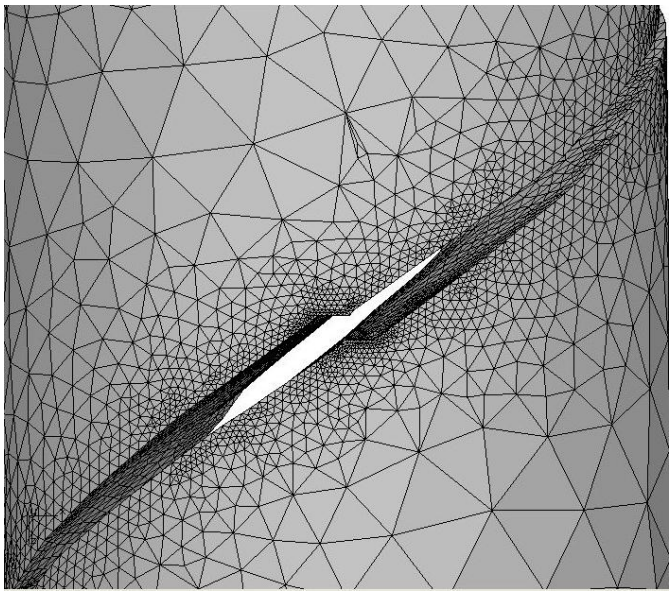
FRANC3D

ANSYS™ Tutorial

ANSYS is a registered
trademark of ANSYS, Inc
www.ansys.com



Version 6

Fracture Analysis Consultants, Inc
www.fracanalysis.com

Revised: November 2011

Table of Contents:

1.0 Introduction 4
2.0 Tutorial 1: Crack Insertion and Growth in a Cube 4
2.1 Step 1: Creating the ANSYS Model 4
Step 1.1: Build ANSYS cube model 5
Step 1.2: Subdivide edges 5
Step 1.3: Add element type 5
Step 1.4: Add material 5
Step 1.5: Mesh the volume 5
Step 1.6: Apply boundary conditions 6
2.2 Step 2: Extract a Portion of the Mesh 7
Step 2.1: Separate element components 7
Step 2.2: Create node component for cut-surface 8
Step 2.3: Save local and global .cdb files 8
2.3 Step 3: Reading Local FE Model into FRANC3D 8
Step 3.1: Reading ANSYS Local FE Model 8
Step 3.2: Selecting the Retained Items in the Local FE Model 9
Step 3.3: Selecting Cut Surface Nodes 11
Step 3.4: Importing and Displaying the Local FE Model 11
2.4 Step 4: Insert a Crack 12
Step 4.1: Selecting Cracks from FRANC3D Menu 12
Step 4.2: Selecting Crack Type 13
Step 4.3: Specify the Crack Size 14
Step 4.4: Specify Crack Location and Orientation 14
Step 4.5: Specify Crack Front Template Parameters 15
Step 4.6: Surface and Volume Meshing of Local Model after the Crack Insertion 16
2.5 Step 5: Static Crack Analysis 17
Step 5.1: Select Static Crack Analysis 17
Step 5.2: Select FE Solver 18
Step 5.3: Select Analysis Options 19
Step 5.4: Merging Local/Global FE Models 20
2.6 Step 6: Compute Stress Intensity Factors 22
Step 6.1: Re-Open FRANC3D restart file 22
Step 6.2: Select Compute SIFs 22
2.7 Step 7: Manual Crack Growth 24
Step 7.1: Select Grow Crack 24
Step 7.2: Specify Growth Rate 25
Step 7.3: Specify Extesnion or Cycles 25
Step 7.4: Specify Fitting and Extrapolation 26
Step 7.5: Specify Crack Front Template 27
2.8 Step 8: Automatic Crack Growth 28
Step 8.1: Open FRANC3D Restart File 28
Step 8.2: Select Crack Growth Analysis 29
Step 8.3: Specify Growth Parameters 29

Step 8.4: Specify Growth Model Data 30
Step 8.5: Specify Fitting and Template Parameters 31
Step 8.6: Specify Extension or Cycle Data 32
Step 8.7: Specify Analysis Code 32
Step 8.8: Specify Analysis Options 33
Step 8.9: Specify Local/Global Model Connection 34
3.0 Tutorial 2: Corner Crack in a Plate, with Crack Face Traction, with Static and Automatic Analysis 36
3.1 Step1: Creating the ANSYS Mesh Model 36
Step 1.1: Define Element Type 36
Step 1.2: Define Material Properties 36
Step 1.3: Define Material Properties 36
Step 1.4: Subdivide Edges 37
Step 1.5: Mesh Volume 37
Step 1.6: Apply Boundary Conditions 38
Step 1.7: Analyze Model in ANSYS 39
Step 1.8: Save Model to .cdb File 39
3.2 Step 2: Reading FE Model into FRANC3D 40
Step 2.1: Read the ANSYS Mesh 40
Step 2.2: Selecting the Retained Items in the Local FE Model 41
3.3 Step 3: Insert a Crack 42
Step 3.1: Select New Flaw Wizard and Flaw Type 42
Step 3.2: Specify Crack Type 42
Step 3.3: Specify Crack Dimensions 44
Step 3.4: Specify Crack Location and Orientation 44
Step 3.5: Specify Crack Front Template 45
3.4 Step 4: Static Crack Analysis 46
Step 4.1: Select Static Crack Analysis 47
Step 4.2: Specify Analysis Code 47
Step 4.3: Specify Analysis Options 48
3.5 Step 5: Compute Stress Intensity Factors 49
Step 5.1: Restart FRANC3D 49
Step 5.2: Select Compute SIFs 49
3.6 Step 6: Crack Analysis with Crack Face Traction 50
Step 6.1: Read the FE Model 51
Step 6.2: Insert a Crack 51
Step 6.3: Select Crack Face Pressure/Traction 51
Step 6.4: Specify Residual Stress Defined on Mesh 51
Step 6.5: Static Crack Analysis 53
Step 6.6: Compute Stress Intensity Factors 54
3.7 Step 7: Comparison of Stress Intensity Factors 55
4.0 Tutorial 3: Center Through-Crack in a Plate Sub-Domain, with Two Crack Fronts, Two Load Cases, and a SIF History 57
4.1 Step 1: Create the uncracked model using ANSYS 57
4.2 Step 2: Crack Insertion and Static Analysis 58
Step 2.1: Read the FE Mesh and Select Retained Data 58

Step 2.2: Crack Insertion 59
Step 2.3: Static Crack Analysis 61
4.3 Step 3: Compute SIFs and Grow Crack Fronts 62
Step 3.1: Compute SIFs 62
Step 3.2: Crack Growth of Two Crack Fronts 63
4.4 Step 4: Automatic Crack Growth Analysis 64
Step 4.1: Run Automatic Crack Growth Analysis 64
Step 4.2: Monitor Analyses 65
4.5 Step 5: SIF History 65
Step 5.1: Create Growth History 65
Step 5.2: Extract SIF History 65
4.6 Step 6: Multiple Load Cases 67
Step 6.1: Restart FRANC3D 67
Step 6.2: Apply Crack Face Pressure/Traction 67
Step 6.3: Run Static Analysis 68
Step 6.4: Compute SIFs 69

1.0 Introduction

This manual contains tutorial examples for FRANC3D Version 6 with ANSYS Version 12 (versions 11 and 13 should work also). The basic capabilities of FRANC3D (and ANSYS) are illustrated by first analyzing a surface crack in a cube. Subsequent tutorial examples build on the first example and describe additional capabilities and features of the software. It is intended that the user perform the operations as they are presented, but you should feel free to experiment and consult the other reference documentation whenever necessary.

Menu and dialog box button selections are indicated by bold text, such as **File**. Model and corresponding file names will be indicated by *italic text.* Window regions and dialog options, fields and labels will be underlined.

2.0 Tutorial 1: Crack Insertion and Growth in a Cube

We start by modeling a surface crack in a cube under far-field tension. First, all the steps needed to create the cube model using ANSYS are briefly described. It is assumed that the user is familiar with ANSYS. Once the model is created in ANSYS, the FRANC3D steps necessary to read the mesh information, insert a crack, rebuild the mesh, perform the ANSYS analysis, and compute stress intensity factors are all described.

2.1 Step 1: Creating the ANSYS Model

First, we create a cube model using ANSYS. We assume that the user knows how to use ANSYS, but we provide enough details in the steps below for a novice user to create the simple cube model. We are using the ANSYS ADPL classic interface. If the user prefers to use the ANSYS Workbench interface, that should also work; the end result is the *.cdb* file that FRANC3D will read.

Step 1.1: Build ANSYS cube model

Start with the ANSYS ADPL graphical user interface and select **Preprocessor, Modeling, Create, Volumes, Block, By Dimensions** and enter -10 and +10 for dimension for the three axes and select **OK**.

Step 1.2: Subdivide edges

The edges can be subdivided; we will use 5 subdivisions on each edge. Select **Preprocessor, Meshing, Size Cntrls, Lines, All Lines** and enter 5 for the NDIV No. of element divisions and select **OK**.

Step 1.3: Add element type

We need to define an element type before we can mesh the volume. We will choose Solid95, which are second order brick elements. Select **Preprocessor, Element Type, Add/Edit/Delete** and then select **Add** on the dialog box. A list of element types is presented. Choose Solid from the Structural/Mass list and then choose Brick 20node 95 (Solid95) and select **OK** and then **Close**.

Step 1.4: Add material

The material properties should be defined next. Select **Preprocessor, Material Props, Material Models** and then select Structural (double click) Linear, Elastic, and then Isotropic. Enter 10000 for the elastic modulus (EX) and 0.3 for the Poisson's ratio (PRXY) in the dialog box and select **OK**. You can then close the **Material Models** dialog box.

Step 1.5: Mesh the volume

The volume can now be meshed using a mapped mesh; this will generate 20-noded brick elements. Select **Preprocessor, Meshing, Mesh, Volume, Mapped,** and then **4 to 6 sided**. Pick the volume; since there is only one volume, you can **Pick All** and the volume will be meshed.

Step 1.6: Apply boundary conditions

Boundary conditions will consist of displacement constraints on the base surface and uniform traction (a negative pressure in ANSYS) on the top surface. The boundary conditions can be applied to the nodes directly. Select **Preprocessor, Loads, Define Loads, Apply, Structural, Displacement,** and then **on Nodes**. Select the Box radio button and then drag a box on the screen such that you collect all the nodes on the bottom surface. Then select **OK** and then choose ALL DOF in the dialog box that will be presented and select **OK**. The constraint symbols should be shown on the model. We repeat this process, choosing **Pressure** instead of **Displacement**, choosing the nodes on the top surface, and entering -1 for the constant pressure.

The resulting model should appear as in Fig 2.1. The symbols for the boundary conditions are displayed attached to the mesh entities.

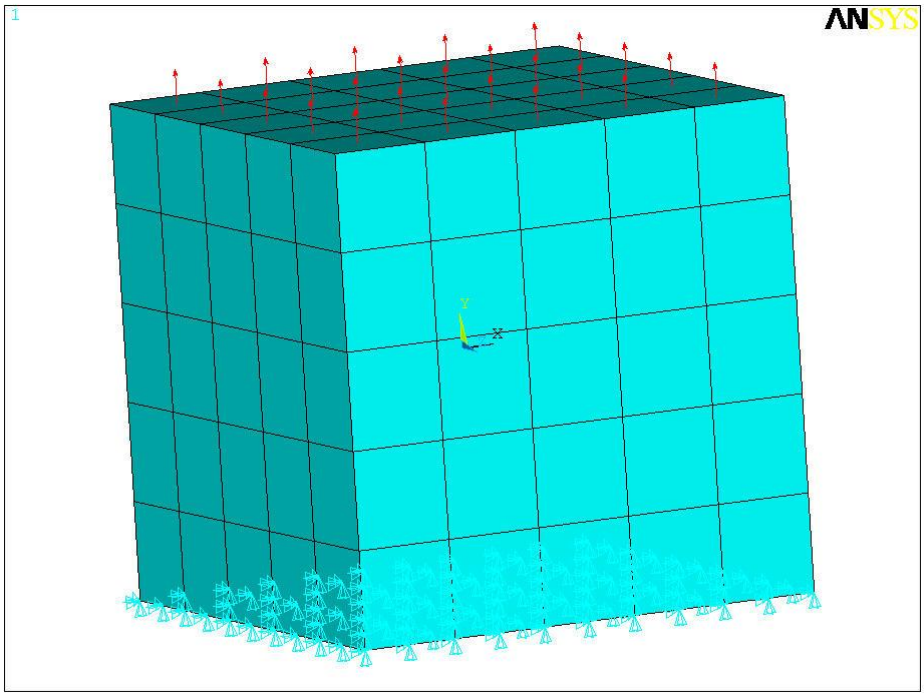


Figure 2.1: ANSYS cube mesh and boundary conditions.

2.2 Step 2: Extract a Portion of the Mesh

We will now extract a small portion of the mesh for fracture analysis. This is not necessary, but it illustrates the process that will be useful for larger models. *If you wish to proceed without dividing the model into pieces, write out the .cdb file for the full model and then jump to Step 3 in Section 2.3; you must retain all the boundary conditions and material properties if you do this.*

Step 2.1: Separate element components

Separate the elements as shown in Fig 2.2. You will need to know or learn how to use the **Select** menu option and know or learn how to create components. The smaller cube will consist of 3x3x3 elements. From the ANSYS menu bar, choose **Select** and **Entities**. In the **Select Entities** dialog, choose Elements and By Num/Pick and use Reselect, which allows you to select only from the current selection. Select **OK** and then begin picking the elements that will make up the 3x3x3 'local' model. When the portion is selected, save it as an element component. From the ANSYS menu bar, choose **Select** and **Comp/Assembly** and **Create Component**. Change the Entity Component is made of to Elements and type in a unique name. Save the inverse selection of elements, using a different component name. Note that you can select 'all' and then unselect the component for the 'local' model to get the inverse selection.

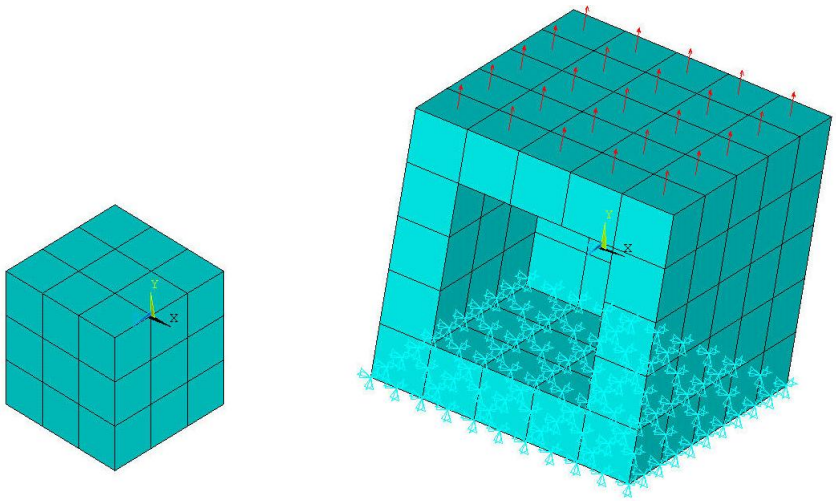


Figure 2.2: ANSYS cube mesh showing extracted portion and remainder.

Step 2.2: Create node component for cut-surface

Select the nodes on the cut surfaces of each component and save a node component. For the 3x3x3 'local' model, name this node component CUT_SURF.

Step 2.3: Save local and global .cdb files

Archive each element component as a separate model, writing the DB information to .cdb files. From the ANSYS Preprocessor menu, select **Archive** and **Write**. The 'global' model, which consists of the exterior elements, will include the boundary conditions and material properties. The 'local' model will include the CUT_SURF node component and FRANC3D will use this information to retain those mesh facets.

2.3 Step 3: Reading Local FE Model into FRANC3D

We start with an existing mesh for FRANC3D. We will use the local 3x3x3 model written in the previous step.

Step 3.1: Reading ANSYS Local FE Model

Start with the FRANC3D graphical user interface, Fig 2.3, and select **File** and **Open**. Switch File Filter in the Open Model File dialog box, Fig 2.3, to Ansys Files (*.cdb) and select the file name for the local model, called *small_cube_cutout.cdb* here, and click **Accept**.

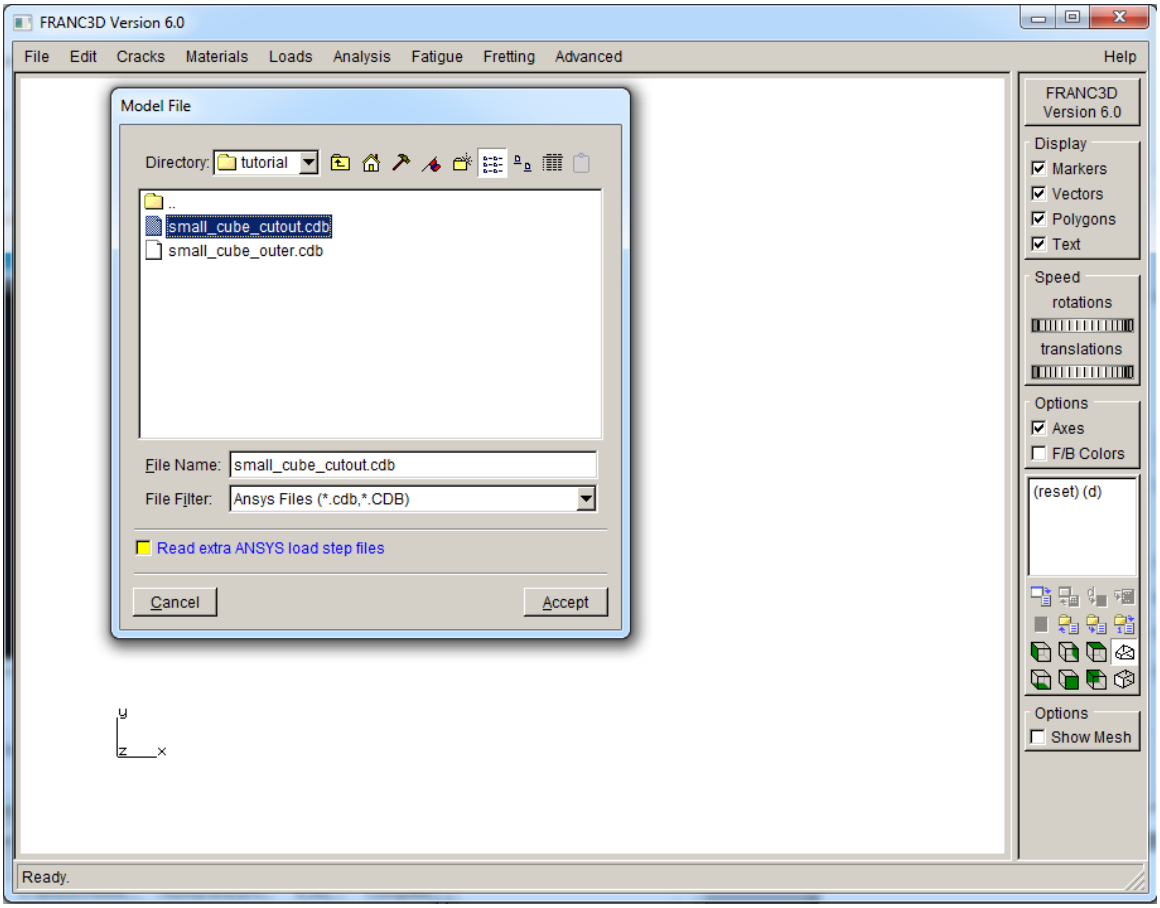


Figure 2.3: FRANC3D graphical user interface.

Step 3.2: Selecting the Retained Items in the Local FE Model

After hitting **Accept** in Step 3.1, the dialog box shown in Fig 2.4 will be displayed, choose the Retain: selected items option. The following wizard panels allow you to choose the data that will be retained from the ANSYS *.cdb* file, in addition to the nodes and elements. The next panel, Fig 2.5, lets you choose to select individual items for each type of data that is present and listed. Choose all materials and choose to select mesh facets groups to be retained. Then select **Next**.

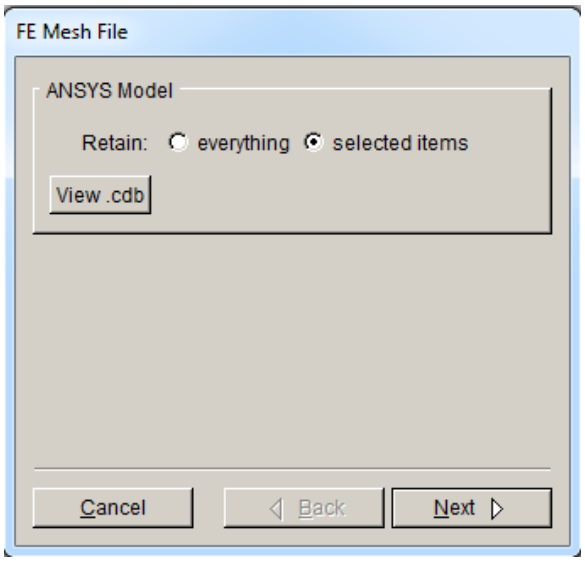


Figure 2.4: ANSYS FE Model retain wizard panel.

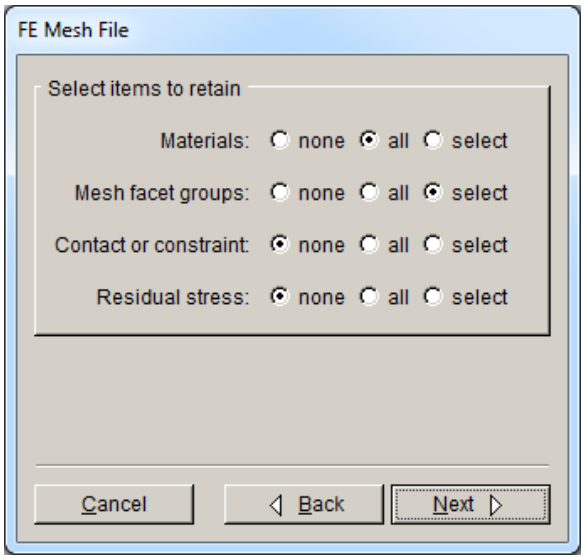


Figure 2.5: Select items to retain wizard panel.

Note that we have retained the material here although it is not actually needed as the global model will have the material data. There are no boundary conditions or other data in this *.cdb* file so no other wizard panels are presented; if there were boundary conditions or coordinate systems, you would have the option of retaining them along with associated mesh facets.

Step 3.3: Selecting Cut Surface Nodes

The next wizard panel, Fig 2.6, lists the node components present in the *.cdb* file; these should be checked if the corresponding mesh faces are to be retained. In this example, we choose to retain the mesh associated with CUT_SURF, which is the set of nodes common to the local and global models. Select **Finish**.

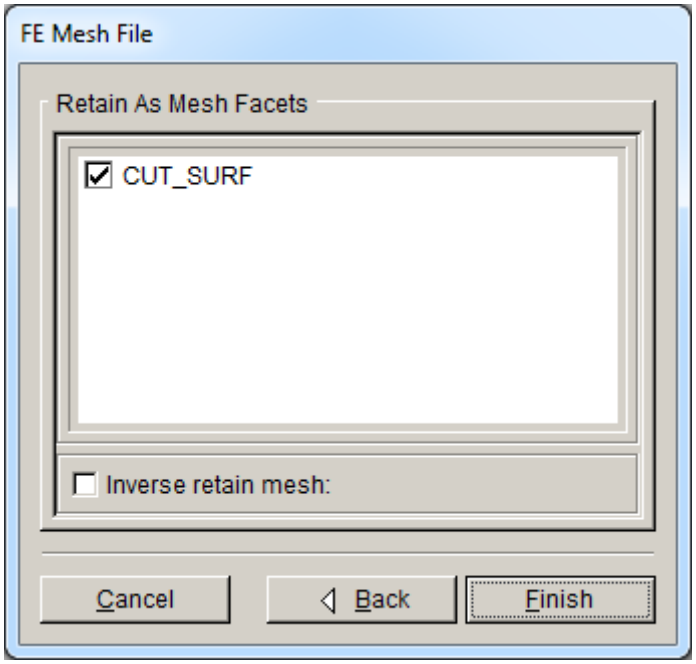


Figure 2.6: Retain mesh facets wizard panel.

Step 3.4: Importing and Displaying the Local FE Model

The model will be imported and displayed in the FRANC3D modeling window, Fig 2.7. You can turn on the surface mesh and manipulate the view (see Section 2.1 of the FRANC3D Reference guide for more details). The model should appear as in Fig 2.7, which shows that the mesh is retained on the cut surfaces.

.

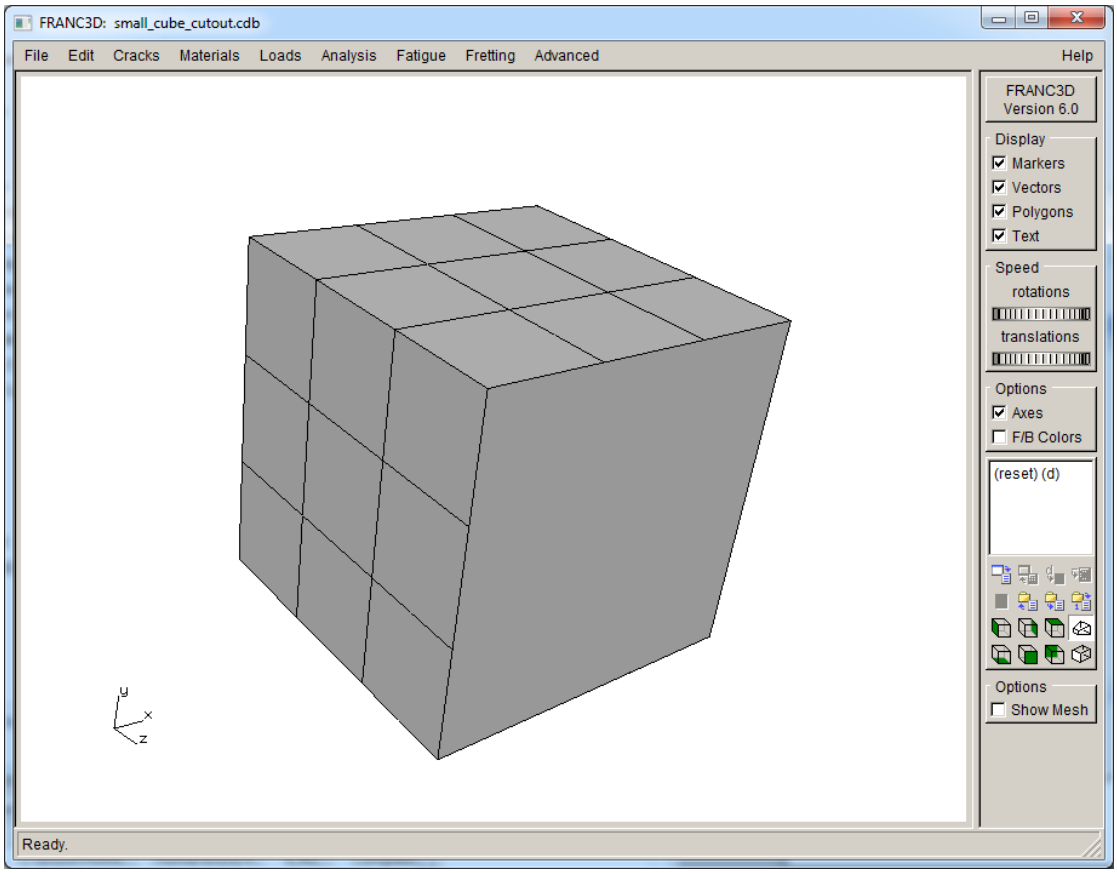


Figure 2.7: Local 3x3x3 ANSYS model converted to FRANC3D with retained mesh faces on the cut_surface.

2.4 Step 4: Insert a Crack

We will now insert a half-penny surface crack into the model.

Step 4.1: Selecting Cracks from FRANC3D Menu

From the FRANC3D menu, select **Cracks** and **New Flaw Wizard**. The first panel of the wizard should appear as in Fig 2.8. The default flaw type is Crack (zero volume flaw) and this is what we want, so select **Next**.

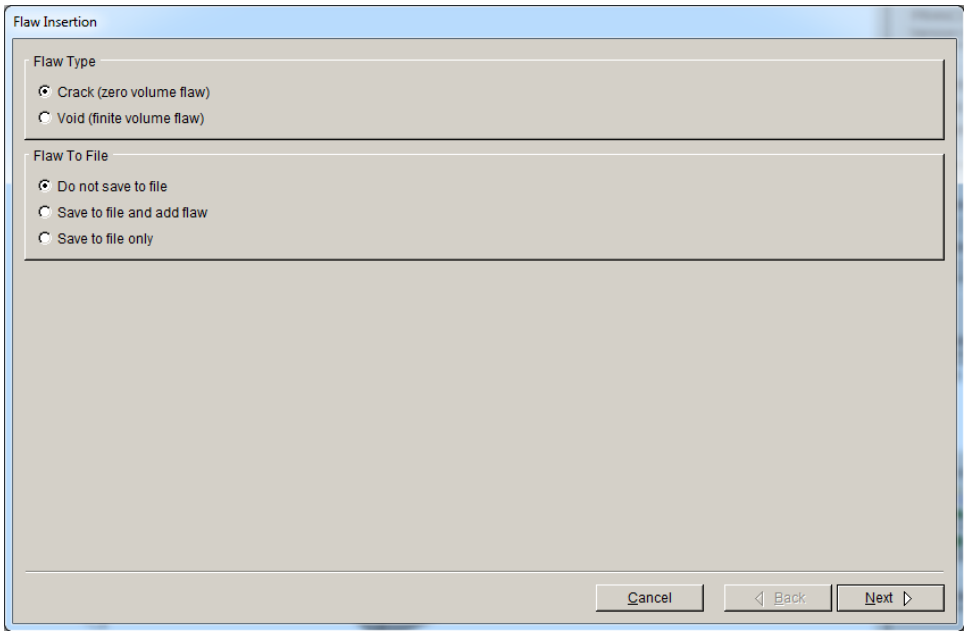


Figure 2.8: New flaw wizard first panel to choose flaw type.

Step 4.2: Selecting Crack Type

The next panel of the wizard, Fig 2.9, allows you to choose the type of crack, either an elliptical, a through-crack, or a user-defined shape. The default shape is the ellipse, which is what we want, so select **Next**.

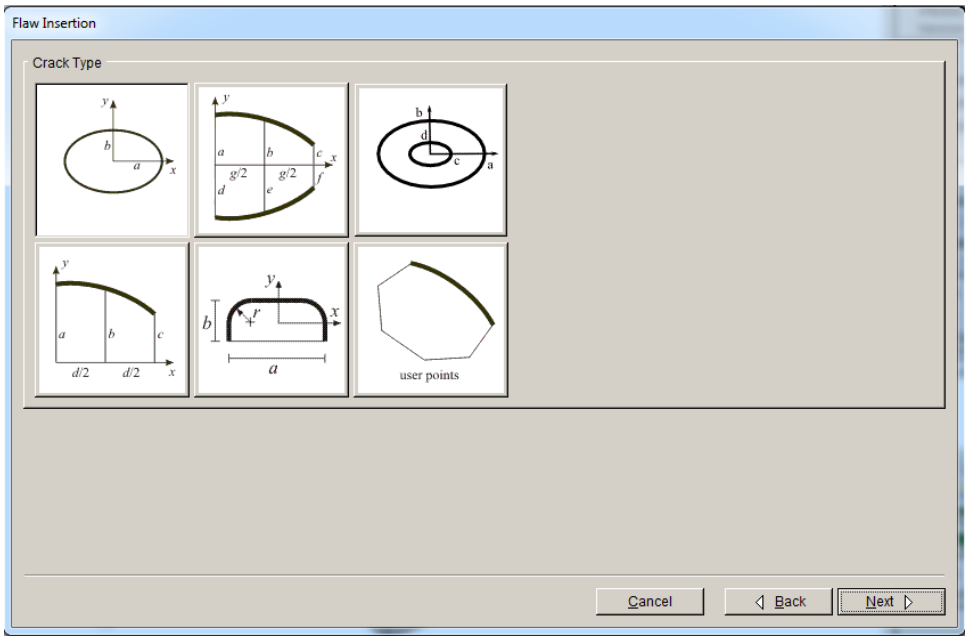


Figure 2.9: Flaw wizard panel to choose crack type.

Step 4.3: Specify the Crack Size

The next panel of the wizard, Fig 2.10, allows us to specify the size of the ellipse. Enter 0.5 for both a and b and select **Next**.

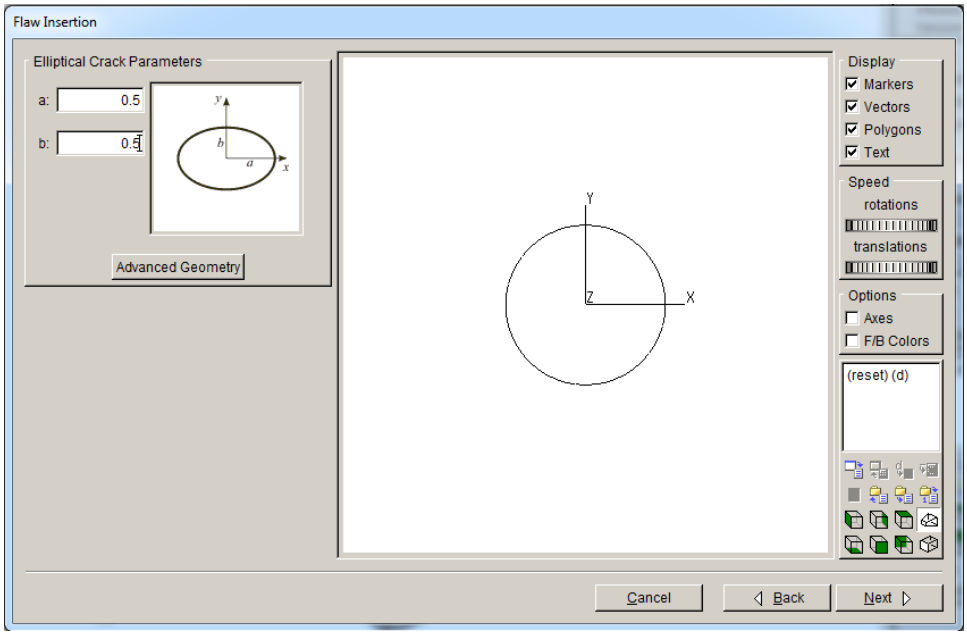


Figure 2.10: Flaw wizard panel to set size of ellipse.

Step 4.4: Specify Crack Location and Orientation

The next panel of the wizard, Fig 2.11, allows us to specify location and orientation of the flaw. Set the Z axis Translation to 10. Enter 90 for the 1st Rotation Angle and set the axis to X. The flaw is displayed along with the model and should appear as in Fig 2.11; select **Next** when ready.

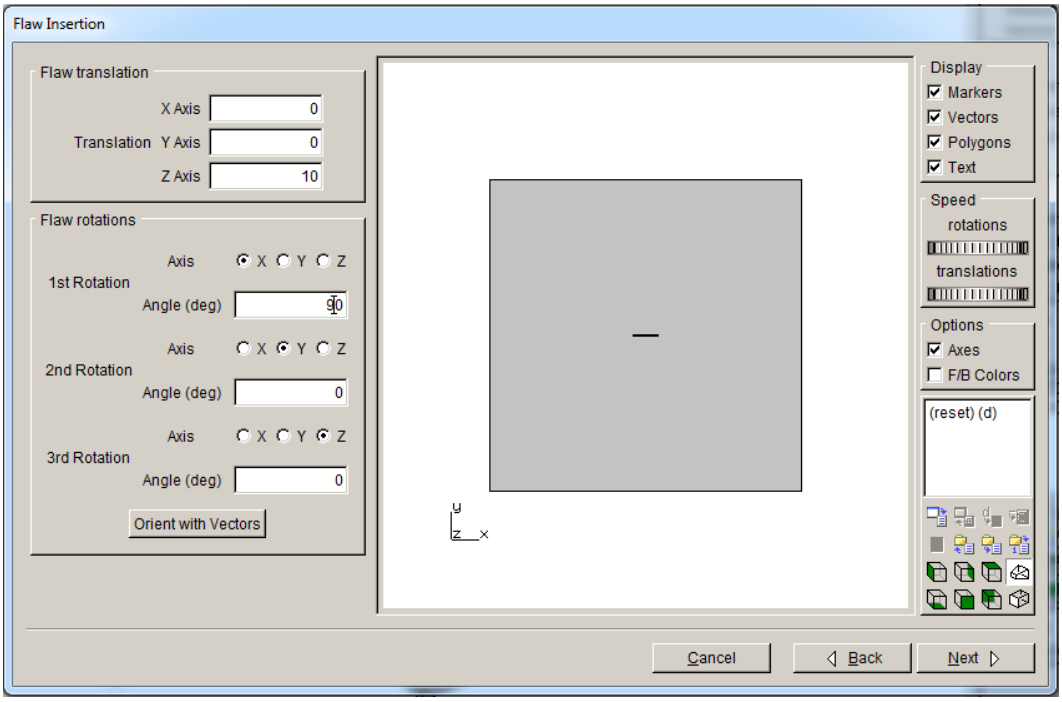


Figure 2.11: Flaw wizard panel to set location and orientation.

Step 4.5: Specify Crack Front Template Parameters

The next panel of the wizard, Fig 2.12, allows us to specify the crack front template parameters. We will leave all values at their defaults; select **Finish** when ready.

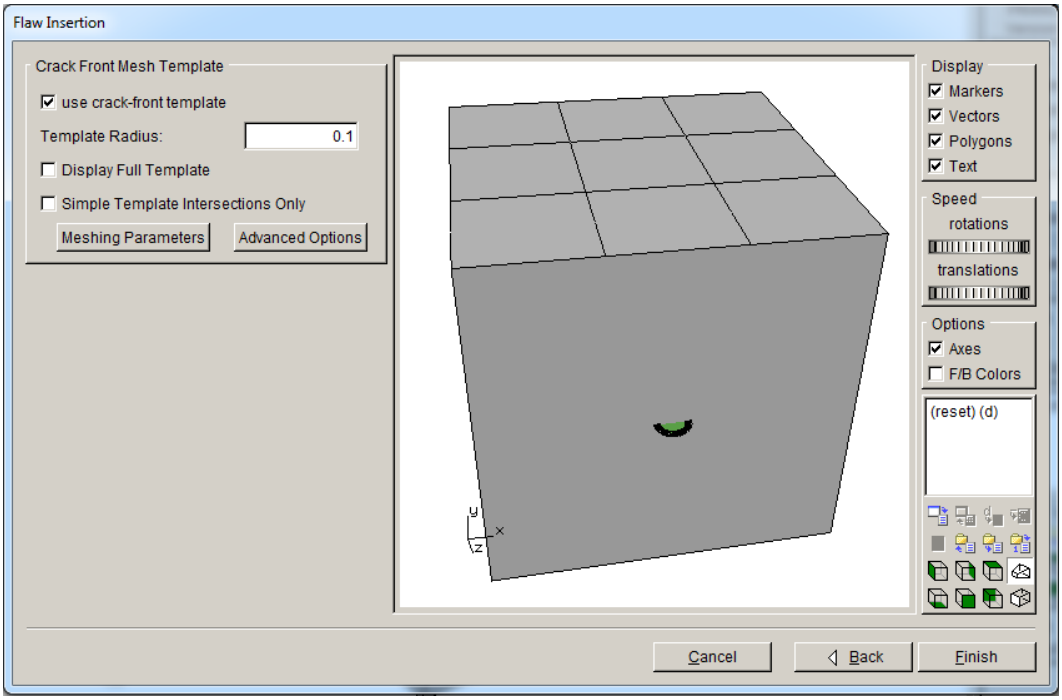


Figure 2.12: Flaw wizard panel to set crack front template parameters.

Step 4.6: Surface and Volume Meshing of Local Model after the Crack Insertion

The program begins the process of inserting the flaw into the original model and then meshes the resulting cracked model. The progress of the operations is displayed on the screen, Fig 2.13. When meshing is complete, the **Flaw Insertion Status** box will disappear and the newly meshed cracked model will be displayed, Fig 2.14.

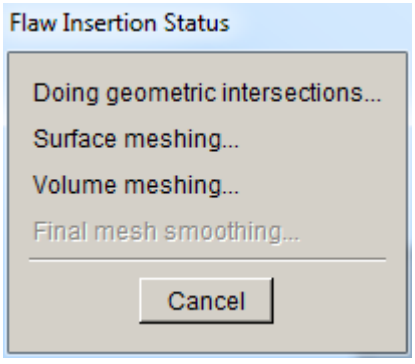


Figure 2.13: Flaw Insertion Status window.

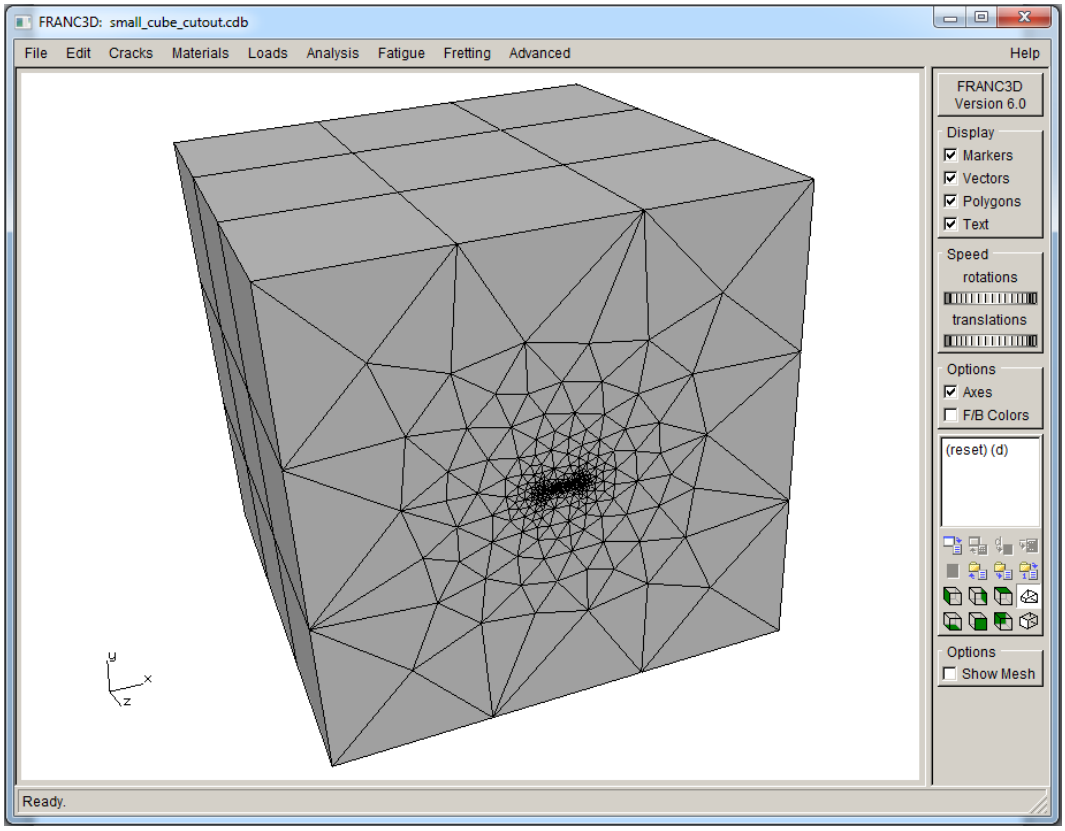


Figure 2.14: Meshed model with crack.

2.5 Step 5: Static Crack Analysis

We will now perform the stress analysis in combination with ANSYS.

Step 5.1: Select Static Crack Analysis

From the FRANC3D menu, select **Analysis** and **Static Crack Analysis**. The first panel of the wizard should appear as in Fig 2.15. We will specify the file name for the FRANC3D database first. We called it *cracked_cube.fdb* here. Note that ANSYS job/file names should be less than 32 characters long. If you exceed 32 characters, ANSYS chops the names and then FRANC3D will have problems reading the files as the names will not match.
Select **Next** once you enter a File Name.

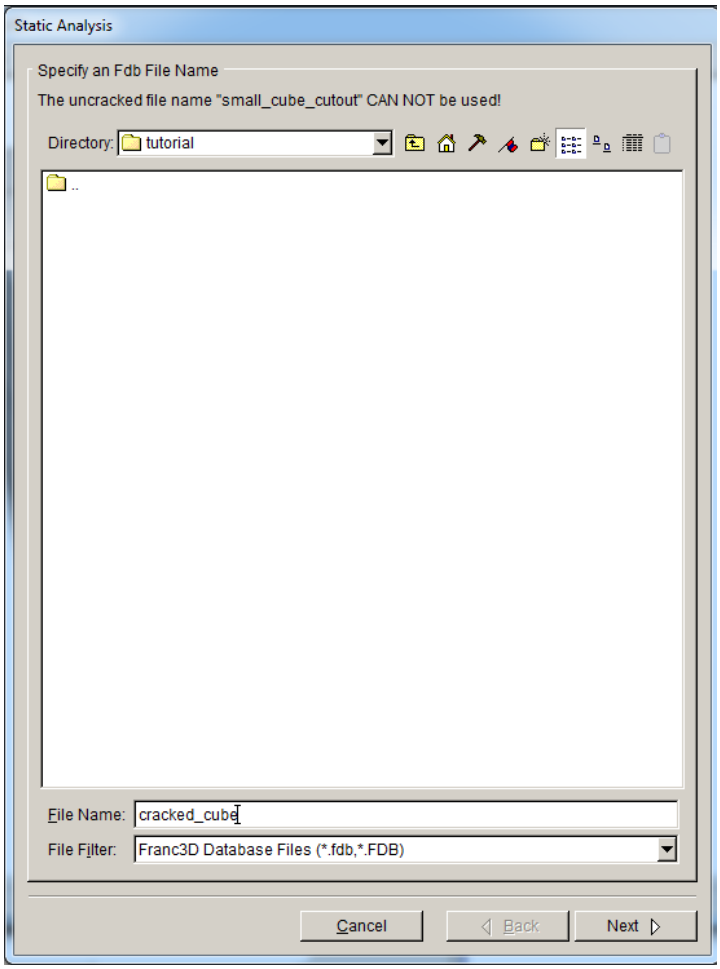


Figure 2.15: ANSYS Static Analysis wizard first panel – File Name.

Step 5.2: Select FE Solver

The next panel of the wizard, Fig 2.16, allows you to specify the solver; choose ANSYS.

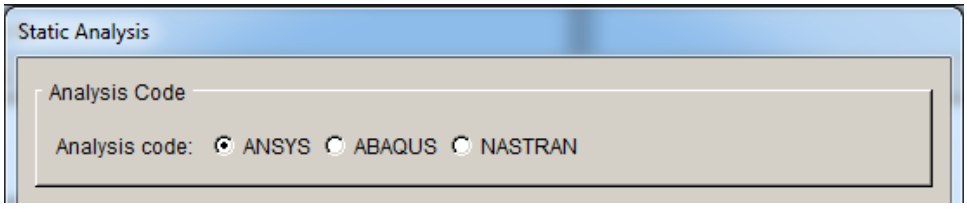


Figure 2.16: Static Analysis wizard second panel – solver.

Step 5.3: Select Analysis Options

The next panel of the wizard, Fig 2.17, allows you to specify the ANSYS output and analysis options. We will connect this model to a global model called *small_cube_outer.cdb*. We want to use all quadratic elements. The ANSYS executable should be defined; this can be saved in the FRANC3D **Preferences** under the **Edit** menu.

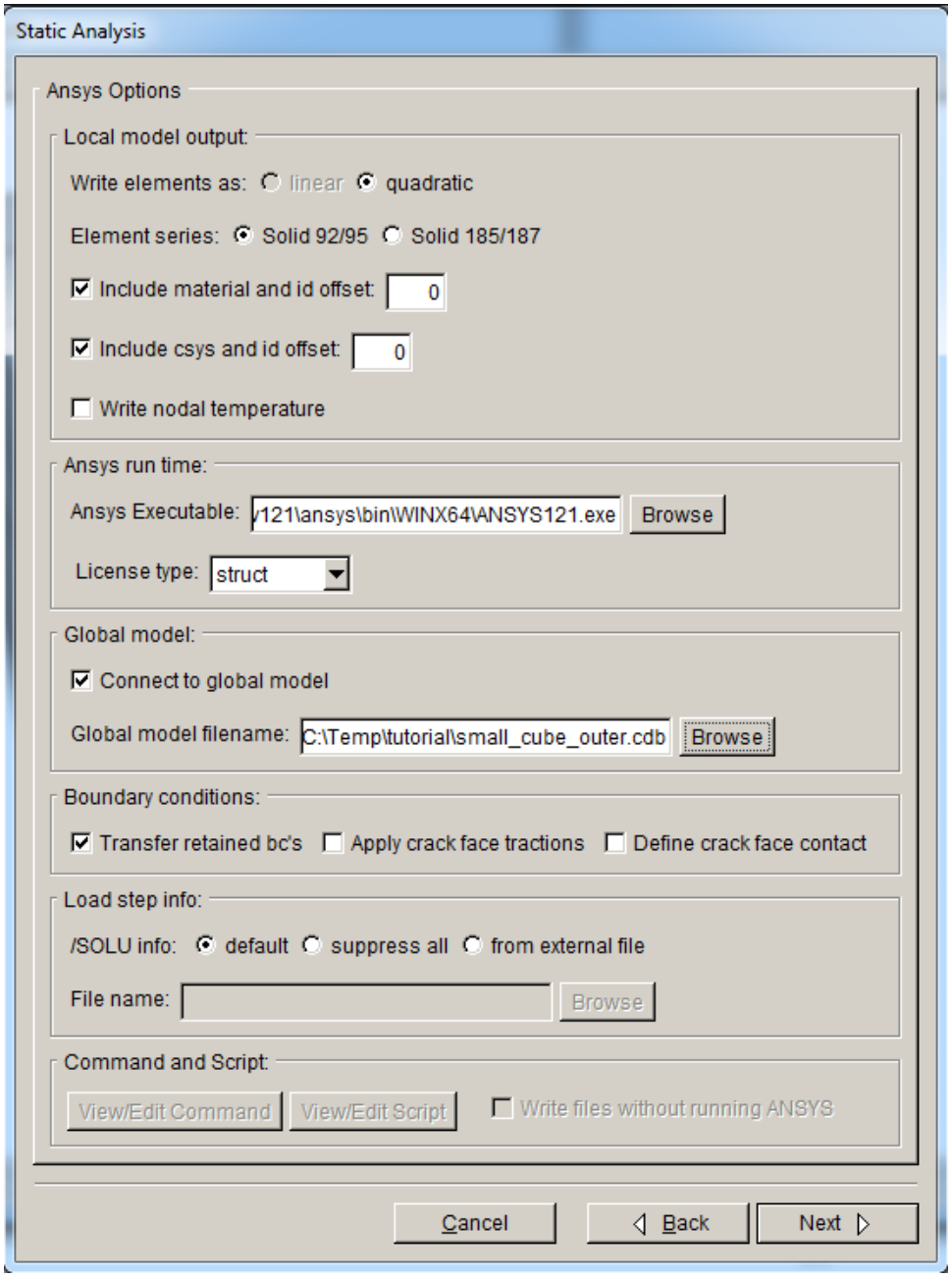

Static Analysis
Ansys Options
Local model output:
Write elements as: linear quadratic
Element series: Solid 92/95 Solid 185/187
Include material and id offset: 0
Include csys and id offset: 0
Write nodal temperature
Ansys run time:
Ansys Executable: v121\ansys\bin\WINX64\ANSYS121.exe Browse
License type: struct
Global model:
Connect to global model
Global model filename: C:\Temp\tutorial\small_cube_outer.cdb Browse
Boundary conditions:
Transfer retained bc's Apply crack face tractions Define crack face contact
Load step info:
/SOLU info: default suppress all from external file
File name: Browse
Command and Script:
View/Edit Command View/Edit Script Write files without running ANSYS
Cancel Back Next

Figure 2.17: Static Analysis wizard third panel – ANSYS analysis and output options.

Step 5.4: Merging Local/Global FE Models

The next panel of the wizard, Fig 2.18, allows you to specify whether the local and global models are combined by merging nodes or by defining constraints or contact conditions. You can specify node component names in the local and global models for nodes that will be merged or you can let the programs (FRANC3D and ANSYS) do the work. FRANC3D creates a *.macro* file of commands that instruct ANSYS to determine nodes to be merged.

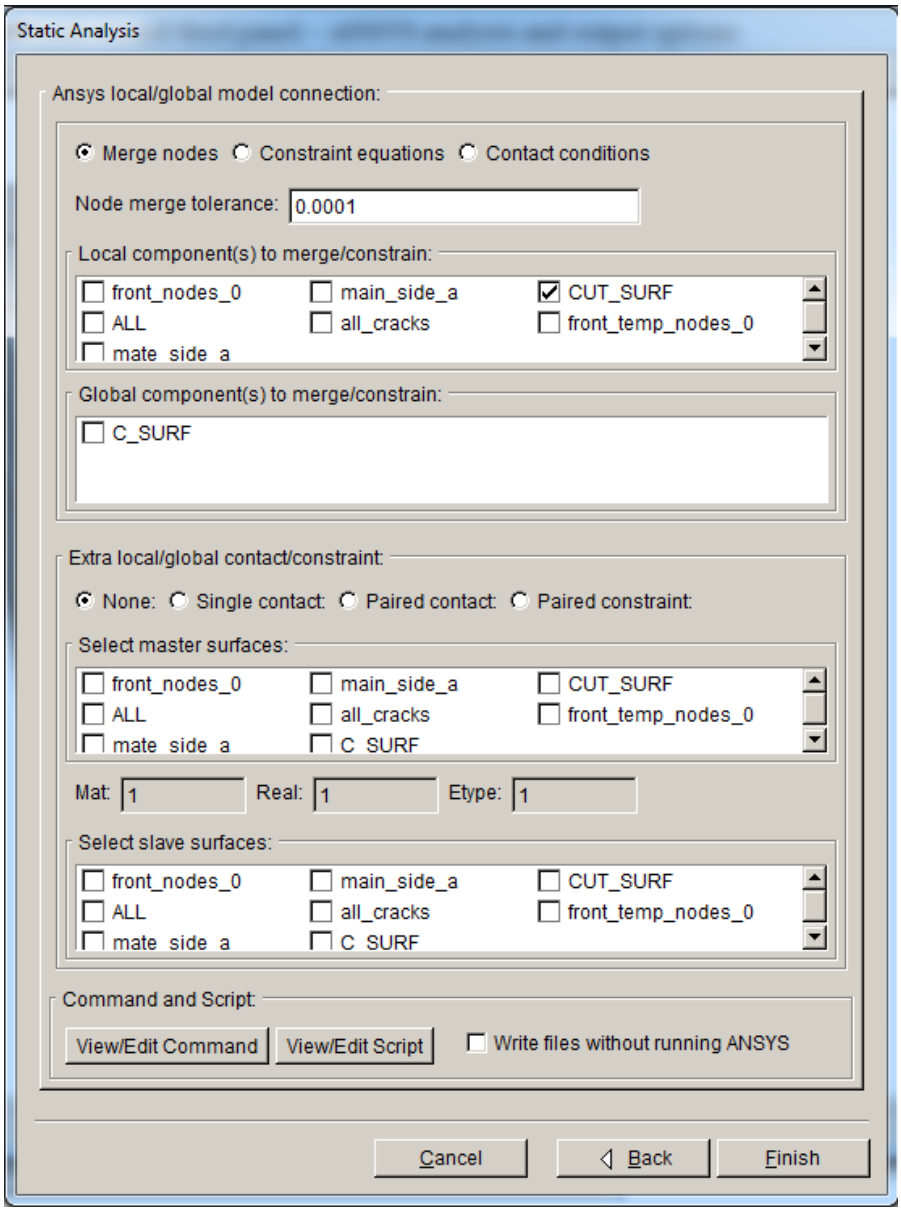


Figure 2.18: Static Analysis wizard fourth panel – ANSYS local/global model connection.

The ANSYS macro listing is shown below for reference. We read the 'global' model followed by the 'local' model to maintain the node numbering for the 'local' model. The two parts are merged together and then the whole model is analyzed. The displacements are output to a *.dtp* file, which is used by FRANC3D to compute SIFs.

```
/BATCH,LIST
/CWD,'C:\Temp\tutorial'
/FILNAME,'cracked_cube',0
/CONFIG,NOELDBW,1
/INPUT,'C:\Temp\tutorial\small_cube_outer.cdb'
/PREP7
/COM, put global model nodes into component
cm,global,node
/COM, collect exterior global model nodes into component
nsel,r,ext
cm,global_ext,node
allsel,all,all
/COM, done with global model for now
fini
/FILNAME,'cracked_cube',0
/INPUT,'C:\Temp\tutorial\cracked_cube','cdb'
/PREP7
cmsel,,CUT_SURF
/COM, add global exterior component
cmsel,a,global_ext
/COM, select all exterior nodes to merge
nummrg,node,0.0001,,sele
/COM, merge exterior nodes of local and global models
nummrg,node,0.0001,,,low
allsel,all,all
save
/COM, select everything and solve
allsel,all,all
/SOLU
eqslv,PCG,1.0e-8
/COM, input solve commands
/INPUT,'C:\Temp\tutorial\cracked_cube','lsm'
/PREP7
cmsel,u,global
/FORMAT,9,G,26,15
/POST1
/GRAPHICS,off
RSYS,0
/COM, reread results
/COM, INRES,ALL
/COM, FILE,'cracked_cube','rst'
/COM, output displacements, temperatures, crack surface pressures to file
/INPUT,'C:\Temp\tutorial\cracked_cube','lsp'
fini
/EXIT,nosav
```

2.6 Step 6: Compute Stress Intensity Factors

We will now compute the stress intensity factors for this crack. If you are able to run ANSYS from FRANC3D, then the model should be open in FRANC3D and the displacement file will be read automatically, and you can skip to Step 6.2.

Step 6.1: Re-Open FRANC3D restart file

If you have closed the model while ANSYS was running, then from the FRANC3D menu, select **File** and **Open**. Choose the *cracked_cube.fdb* file and select **OK.** FRANC3D will automatically read the ANSYS results *.dtp* file if it exists.

Step 6.2: Select Compute SIFs

From the FRANC3D menu, select **Cracks** and **Compute SIFs**. The Stress Intensity Factor wizard is displayed, Fig 2.19. You should use the M-Integral, but you can check that the Displacement Correlation results are similar. There are no thermal or crack face traction terms. When you select **Finish**, the SIFs Plot dialog is displayed, Fig 2.20. You can view the three stress intensity factor (SIF) modes and export the data.

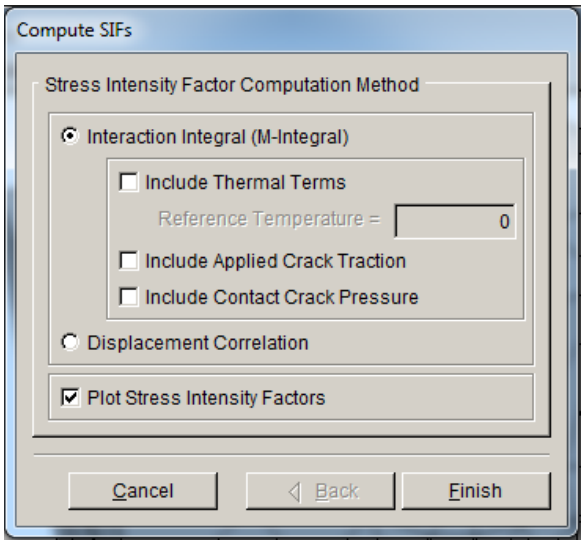


Figure 2.19: Compute SIFs panel.

.

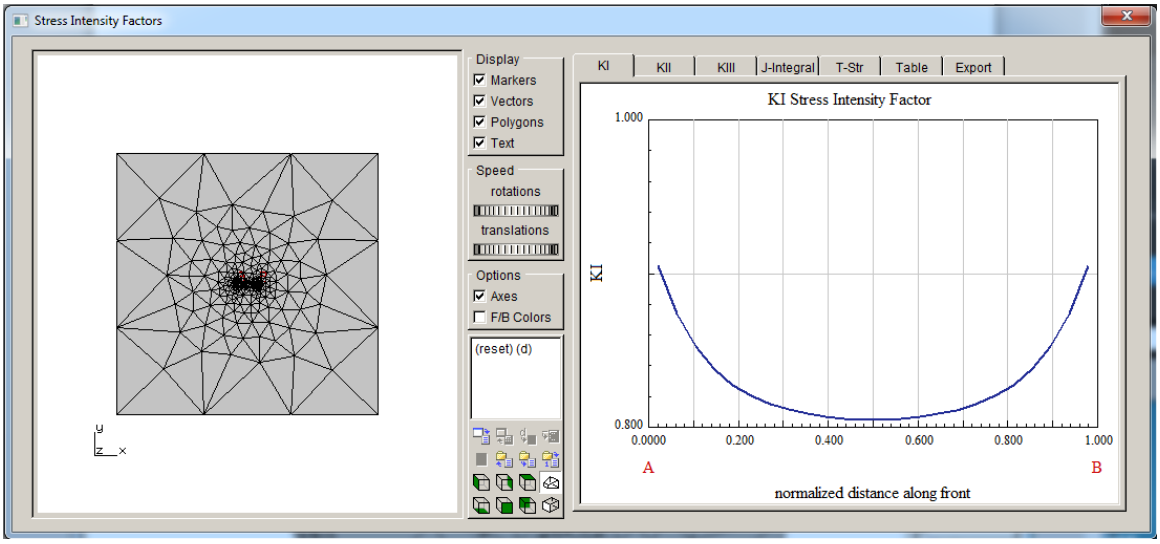


Figure 2.20: Stress Intensity Factor dialog.

The SIF values are computed at the element midpoints along the crack front using the M-integral. These values can be compared with SIFs from the displacement correlation technique, Fig 2.21. The displacement correlation values are slightly lower and the curve is not as smooth.

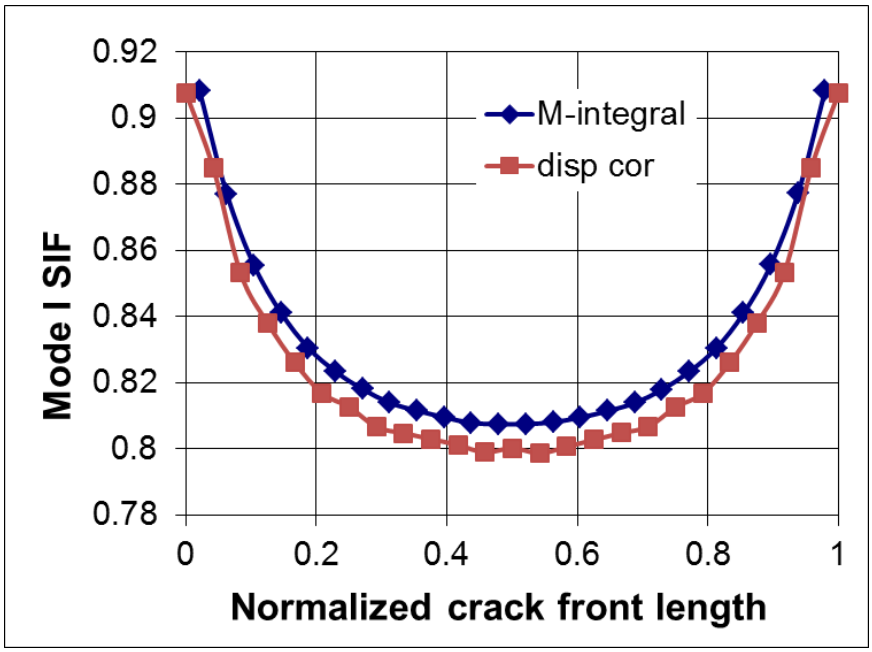


Figure 2.21: Mode I SIFs from FRANC3D using M-integral and displacement correlation.

2.7 Step 7: Manual Crack Growth

We can manually propagate the crack at this stage, and we should at least examine the predicted crack growth to determine suitable parameters for fitting and extrapolation before proceeding with the automated crack growth in Section 2.8.

Step 7.1: Select Grow Crack

From the FRANC3D menu, select **Cracks** and **Grow Crack**. The Crack Growth wizard is displayed, Fig 2.22. We can choose Quasi-Static or Fatigue growth type; we leave all the defaults in this case, and select **Next**.

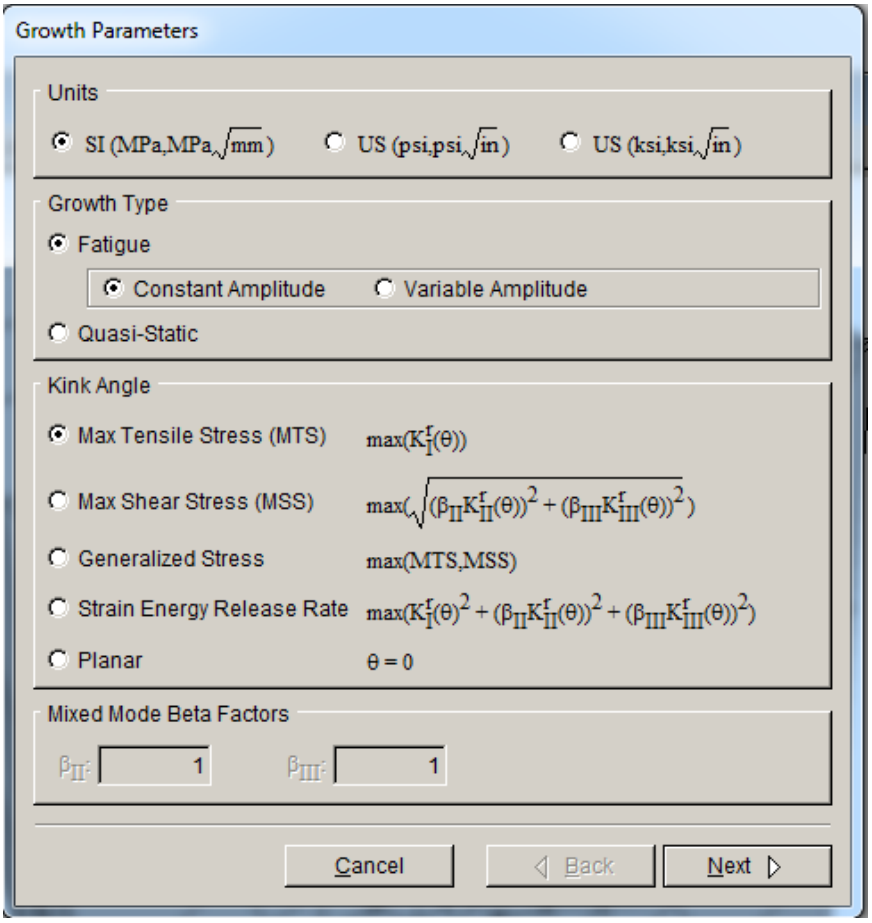


Figure 2.22: Crack Growth wizard – first panel.

Step 7.2: Specify Growth Rate

The second panel of the Crack Growth wizard, Fig 2.23, allows you to specify the growth rate model data. We will use the Paris model and set C to 1e-10 and leave n at 2. Select **Next**.

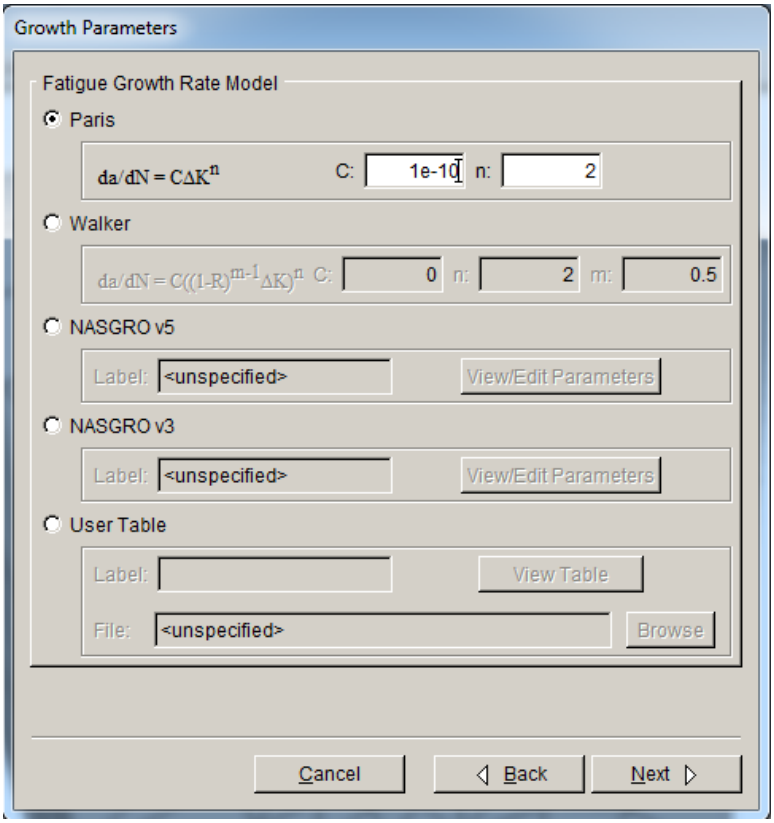


Figure 2.23: Crack Growth wizard – second panel.

Step 7.3: Specify Extesnion or Cycles

The third panel of the Crack Growth wizard, Fig 2.24, allows you to specify whether you will grow the crack based on a median extension or a number of cycles. We use a median extension here and select **Next**.

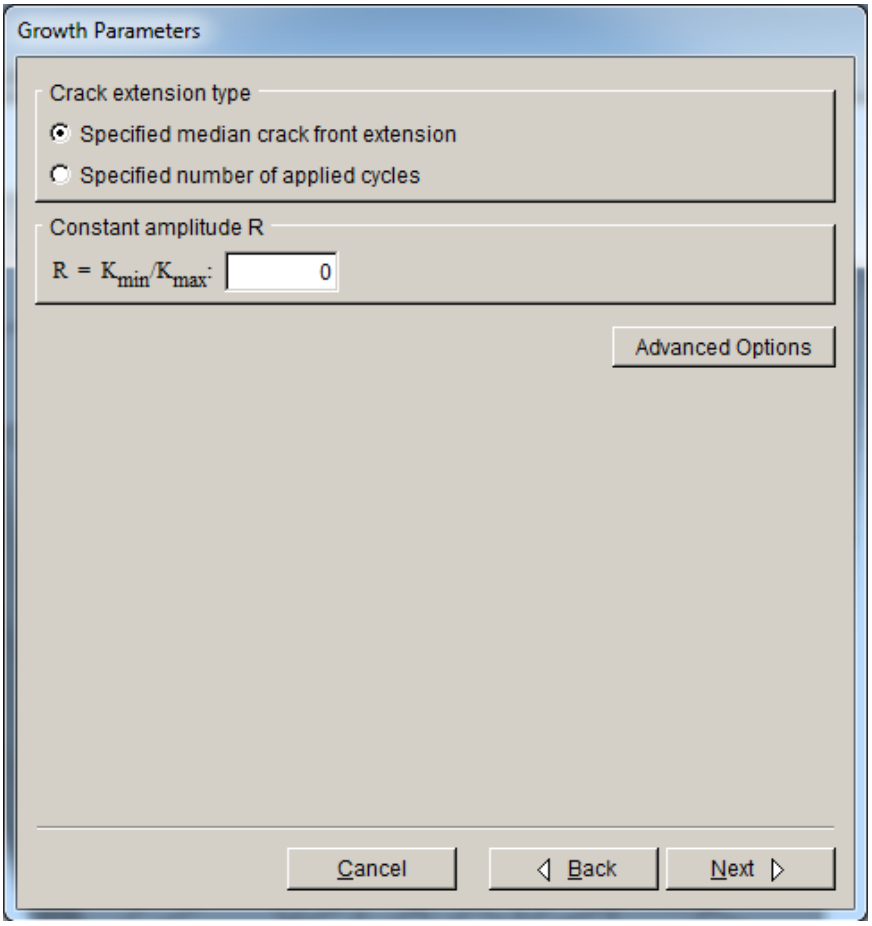


Figure 2.24: Crack Growth wizard – third panel.

Step 7.4: Specify Fitting and Extrapolation

The fourth panel of the Crack Growth wizard, Fig 2.25, allows you to specify a value for median extension as well as the fitting and extrapolation parameters. We specify a median extension of 0.1 and use a fixed 3rd order polynomial with 3% extrapolation on both ends to ensure the fitted end points fall outside the model. Select **Next**.

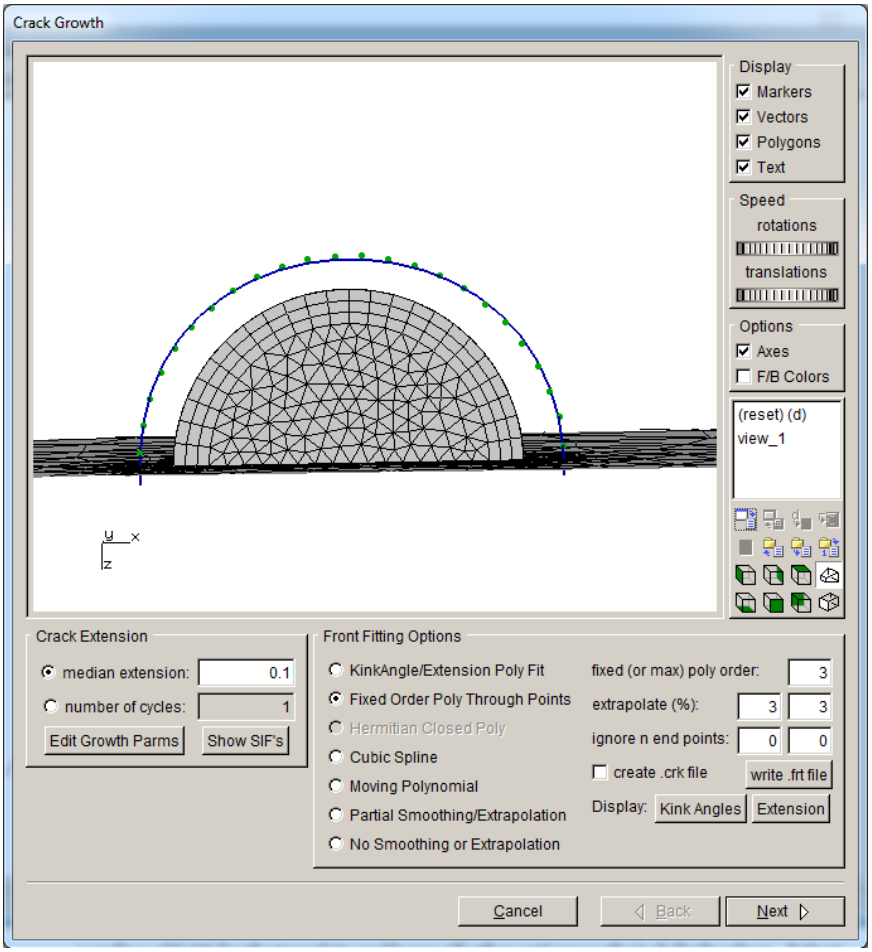


Figure 2.25: Crack Growth wizard – fourth panel.

Step 7.5: Specify Crack Front Template

The final panel, Fig 2.26, allows you to specify the crack front mesh template parameters. We set the template radius to 0.06. Select **Next** to proceed with growing the crack and remeshing.

Once the remeshing is completed, another **Static Crack Analysis** can be performed. Alternatively, one can proceed to the automatic crack growth described in the next section.

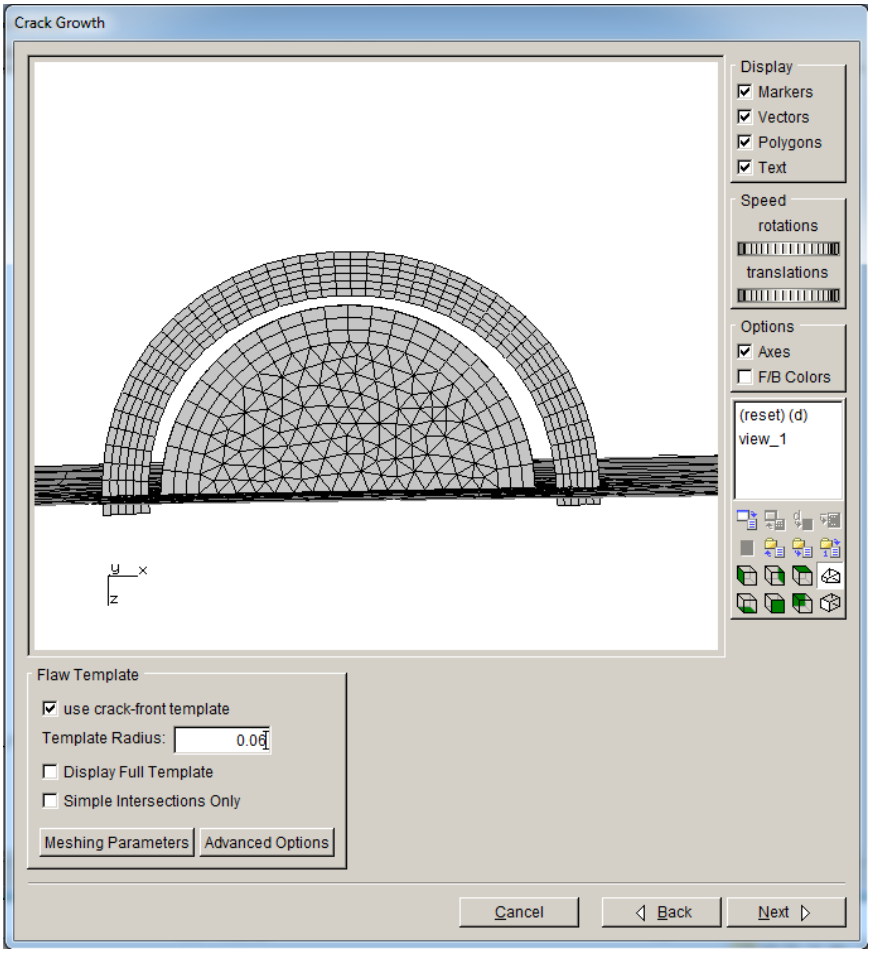


Figure 2.26: Crack Growth wizard – final panel.

2.8 Step 8: Automatic Crack Growth

This section describes the steps taken to do automatic crack growth starting from the initial crack model. We will start with an existing FRANC3D model. We will use the model created in Sections 2.2 and 2.3.

Step 8.1: Open FRANC3D Restart File

Start with the FRANC3D graphical user interface (see Fig 2.3) and select **File** and **Open** and choose the file name specified in Section 2.3, called *cracked_cube.fdb* here. Click **Accept**. The model will be read into FRANC3D (along with the results files that were created in

Section 2.3). We will ignore the fact that we already analyzed and propagated the initial crack for now and proceed with setting up the automatic crack growth analysis.

Step 8.2: Select Crack Growth Analysis

From the FRANC3D menu, select **Analysis** and **Crack Growth Analysis**. The first panel of the wizard should appear as in Fig 2.27; it allows you to choose the method for computing SIFs. We will leave all the default values. Select **Next** to display the second panel.

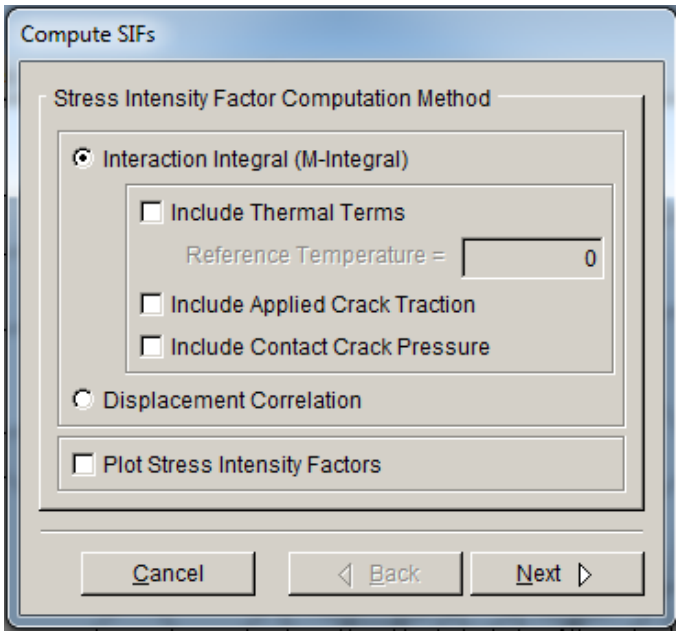


Figure 2.27: Crack Growth Analysis wizard – first panel.

Step 8.3: Specify Growth Parameters

The second panel of the wizard should appear as in Fig 2.28. We set the growth type to Quasi-Static for simplicity. All other values are left as defaults; select **Next**.

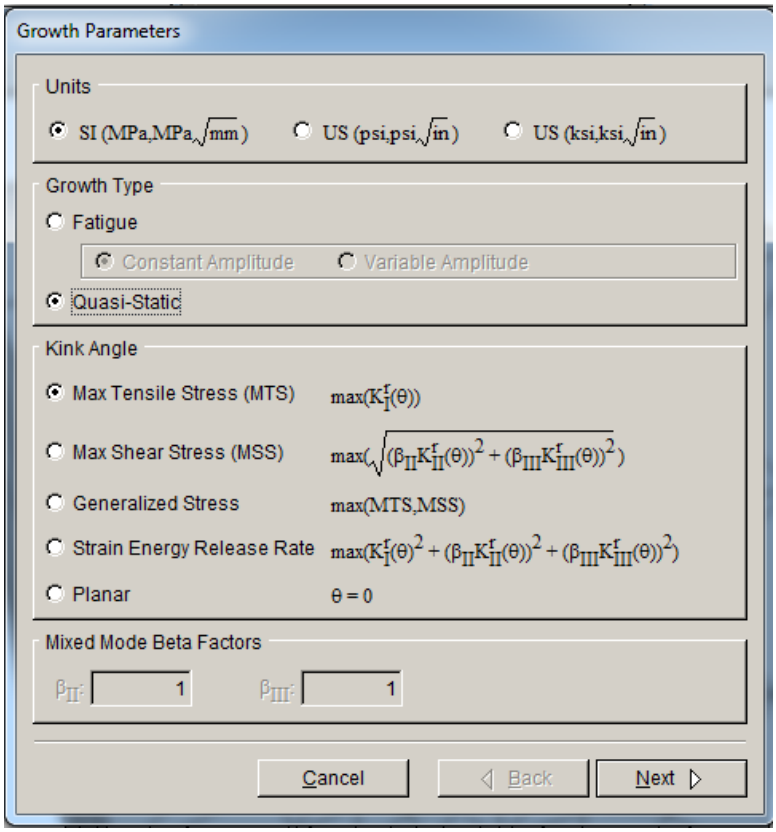


Figure 2.28: Crack Growth Analysis wizard – second panel.

Step 8.4: Specify Growth Model Data

The third panel of the wizard should appear as in Fig 2.29. We set the value of n to 2 for the power-law crack growth model and then select **Next**.

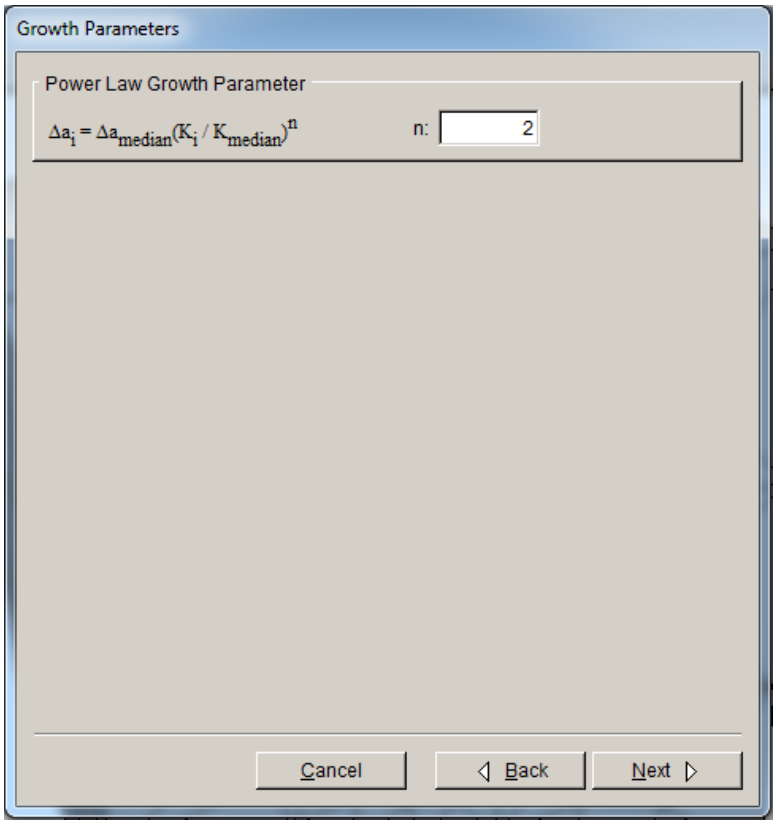


Figure 2.29: Crack Growth Analysis wizard – third panel.

Step 8.5: Specify Fitting and Template Parameters

The fourth panel of the wizard should appear as in Fig 2.30. We set the value for the template radius to 0.06. The extrapolation could be increased from 3 to 5%, but 3% should suffice for the first 5 steps that we will run here. Select **Next**.

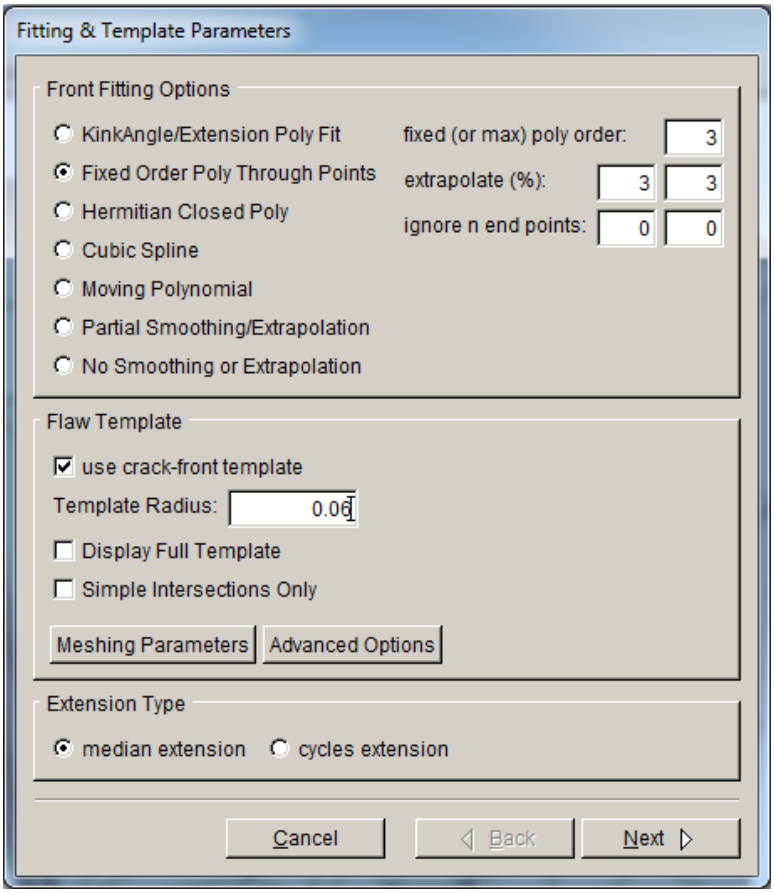


Figure 2.30: Crack Growth Analysis wizard – fourth panel.

Step 8.6: Specify Extension or Cycle Data

The fifth panel of the wizard should appear as in Fig 2.31. We will try growing the crack for 5 steps using a Constant Median Crack Growth Increment of 0.1. Select **Next**.

Step 8.7: Specify Analysis Code

The sixth panel of the wizard should appear as in Fig 2.32. We will use ANSYS and the Current crack growth step is 1 as we are starting from the initial crack. This process will reanalyze the initial crack and name the files as *cracked_cube_STEP_001*. Typically we would analyze the initial crack using a **Static Analysis**, grow the crack using **Grow Crack**, and then start the automatic **Crack Growth Analysis**. The user can choose whether to start the numbering from *_STEP_001* or other. Subsequent file names will have their step number incremented as the automatic analysis proceeds.

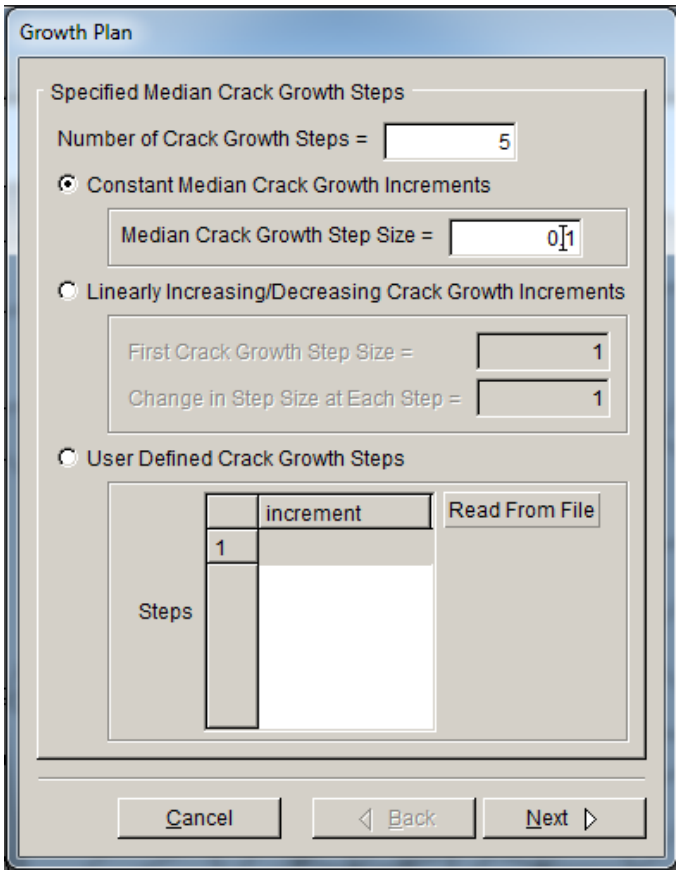


Figure 2.31: Crack Growth Analysis wizard – fifth panel.

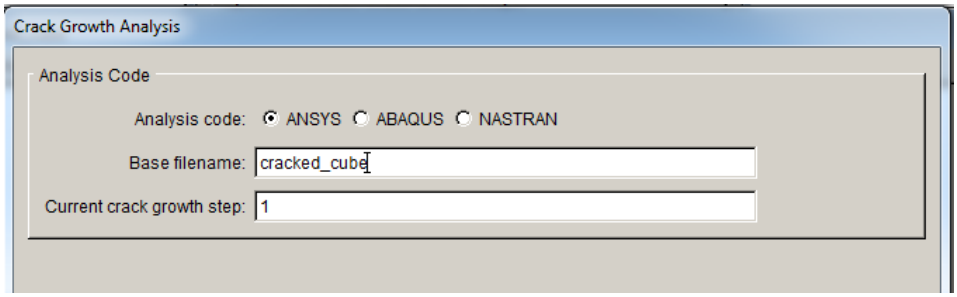


Figure 2.32: Crack Growth Analysis wizard – sixth panel.

Step 8.8: Specify Analysis Options

The seventh panel of the wizard should appear as in Fig 2.33. Some of the options will be specific to your site; we are using ANSYS Version 12. The global model in our case is

called *small_cube_outer.cdb*. We will transfer all the boundary conditions from the global model to the combined model, so leave the Transfer all retained bc's checked. Click **Next**.

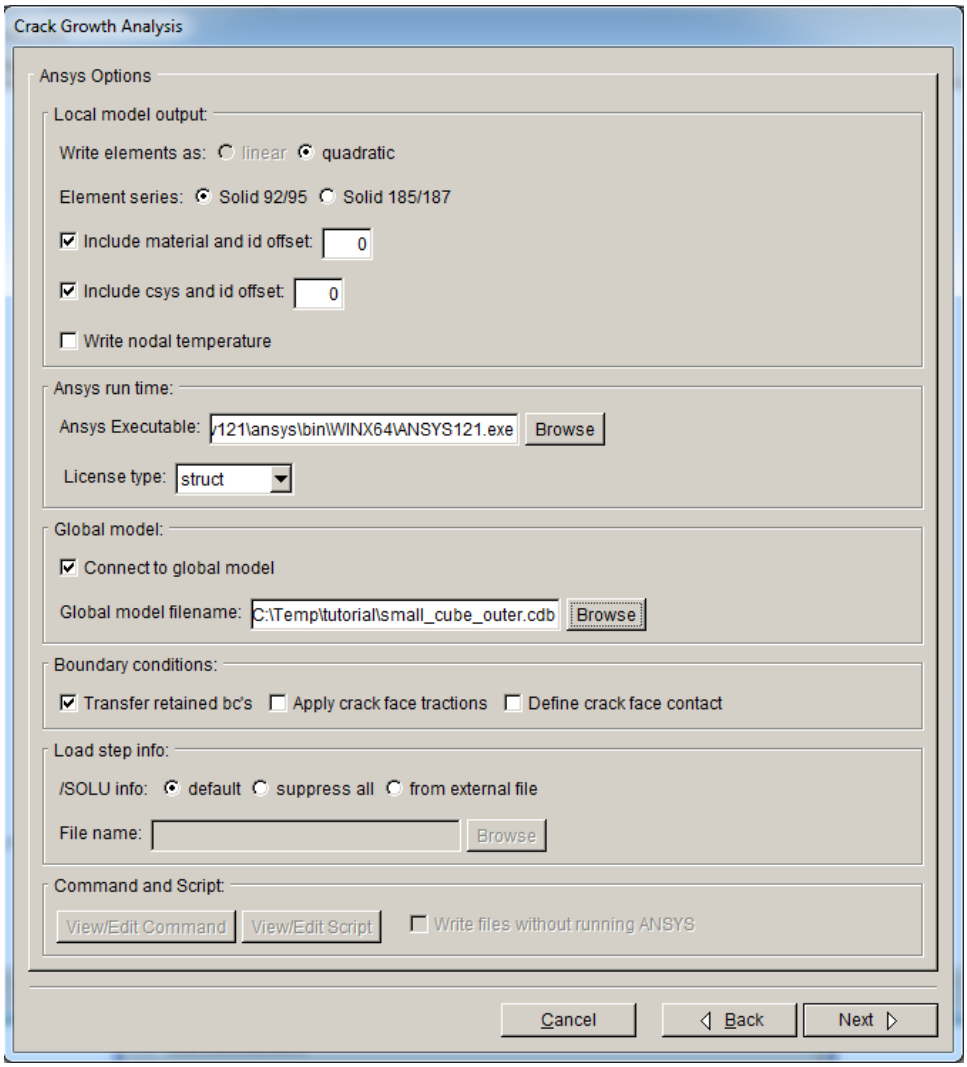

Crack Growth Analysis
Ansys Options
Local model output:
Write elements as: linear quadratic
Element series: Solid 92/95 Solid 185/187
Include material and id offset: 0
Include csys and id offset: 0
Write nodal temperature
Ansys run time:
Ansys Executable: v121\ansys\bin\WINX64\ANSYS121.exe Browse
License type: struct
Global model:
Connect to global model
Global model filename: C:\Temp\tutorial\small_cube_outer.cdb Browse
Boundary conditions:
Transfer retained bc's Apply crack face tractions Define crack face contact
Load step info:
/SOLU info: default suppress all from external file
File name: Browse
Command and Script:
View/Edit Command View/Edit Script Write files without running ANSYS
Cancel Back Next

Figure 2.33: Crack Growth Analysis wizard – seventh panel.

Step 8.9: Specify Local/Global Model Connection

The final wizard panel, Fig 2.34, allows you to choose how the local and global models will be connected. In this case, we will merge the nodes on the cut surfaces. If you do not specify the CUT_SURF local model component name, ANSYS will try to merge any nodes

within the given tolerance, except for the crack nodes. Click **Finish** when you are ready to start the automatic crack growth.

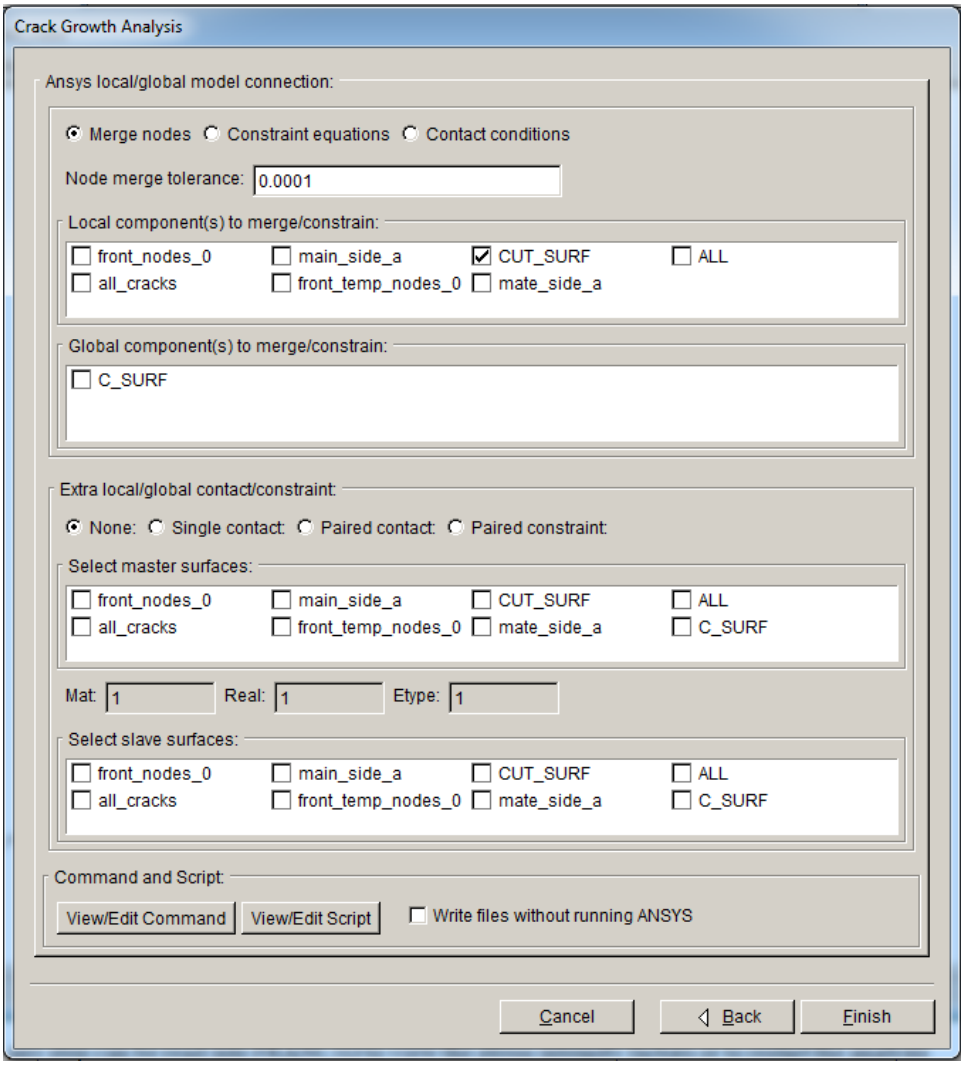


Figure 2.34: Crack Growth Analysis wizard – final panel.

FRANC3D will save the *fdb/cdb* files with the name *small_cube_cutout_STEP_001* for the first crack model and then ANSYS will start in the background. If the analyses stop at any stage, they can be restarted from the last crack step. All of the *_STEP_#* files are retained. The model for any step can be read into FRANC3D to view the stress intensity factors or to restart the analysis with a modified crack growth increment (for example). We will illustrate SIF history extraction and fatigue life computations in Section 4.5.

3.0 Tutorial 2: Corner Crack in a Plate, with Crack Face Traction, with Static and Automatic Analysis

The second tutorial example illustrates the principal of superposition for computing stress intensity factors. We use a simple rectangular bar and compare stress intensity factors for a crack using far-field applied displacement versus a crack with crack face tractions where the tractions are obtained from the stress in the uncracked bar under the far-field loads.

3.1 Step1: Creating the ANSYS Mesh Model

First, we create a rectangular bar model using ANSYS. We assume that the user knows how to use ANSYS, but we provide enough details in the steps below for a novice user to create the model.

Step 1.1: Define Element Type

Start with the ANSYS ADPL Classic graphical user interface and select **Preprocessor, Element Type, Add/Edit/Delete, Block,** and select **Add** in the dialog box that appears. Choose Solid and Brick 8 Node 45 and select **OK** and then **Close**.

Step 1.2: Define Material Properties

The material properties should be defined next. Select **Material Props, Material Models** and then select Structural (double click) Linear, Elastic, and then Isotropic. Enter 10000 for the elastic modulus (EX) and 0.3 for the Poisson's ratio (PRXY) in the dialog box and select **OK**. You can then close the Material Models dialog box.

Step 1.3: Define Material Properties

Next we create the model geometry; select **Modeling, Create, Volumes, Block, By Dimensions** and enter 0 and 0.5 for the x1,x2 coordinates, enter 0 and 1.0 for the y1,y2

coordinates, and enter 0 and 0.25 for the z1,z2 coordinates, and then select **OK**. The model geometry should appear as in Fig 3.1.

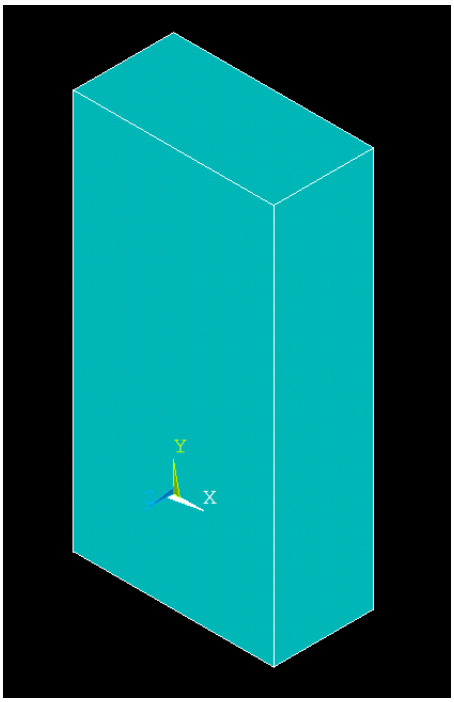


Figure 3.1: Model geometry in ANSYS.

Step 1.4: Subdivide Edges

The edges can be subdivided; we will use 10 subdivisions on all short edges along the z-axis. From **Preprocessor,** select **Meshing, Size Cntrls, Manual Size, Lines, Picked Lines** and pick the four lines and select OK. Enter 10 for the NDIV No. of element divisions and select **OK**. We will use 20 subdivisions on all edges along the x-axis; follow the procedure above. We will use 40 subdivisions on all edges along the y-axis; follow the procedure above.

Step 1.5: Mesh Volume

The volume can now be meshed using a mapped mesh; this will generate 8-noded brick elements. Select **Preprocessor, Meshing, Mesh, Mapped,** and then **4 to 6 sided**. Pick the volume; since there is only one volume, you can Pick All and the volume will be meshed.

Step 1.6: Apply Boundary Conditions

Boundary conditions will consist of displacement constraints on the base surface and applied displacement on the top surface. The boundary conditions can be applied to the geometric areas; they will be transferred to the nodes automatically. Select **Preprocessor, Loads, Define Loads, Apply, Structural, Displacement,** and then **on Areas**. Select the area for the bottom surface. Then select **OK** and then choose UY in the dialog box and select **OK**. The constraint symbol will be shown on the model. We repeat this process to apply displacement on the top surface, entering 0.01 for the constant displacement value in the y-direction. We need to fix the model against x and z motion also. We apply displacement constraint in the x-direction by fixing the line along the z-axis starting from the origin. We fix the key-point at the origin in the z-direction.

The resulting model should appear as in Fig 3.2. The symbols for the boundary conditions are displayed attached to the geometric entities.

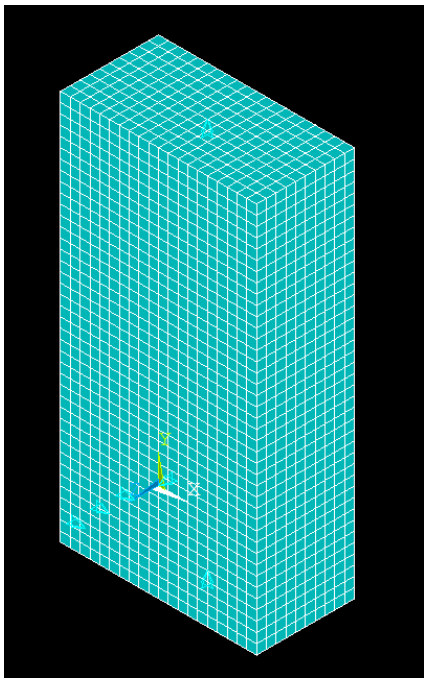


Figure 3.2: Mesh and boundary conditions for ANSYS model.

Step 1.7: Analyze Model in ANSYS

Solve the model using the linear static solver options in ANSYS. Then proceed to **PostProcessing** and make sure the deformed shape is correct (see Fig 3.3). We will list the stress at this point and save the information to a file. First, change the format for the stress listing using the command: /format,,g,20,12 Make sure that **PowerGraphics** is turned off. Then issue the command: prnsol,s Once the listing is displayed, save the data to a file: *rectangular_bar.str*. Save the ANSYS model as *rectangular_bar.db* so that we can resume if needed.

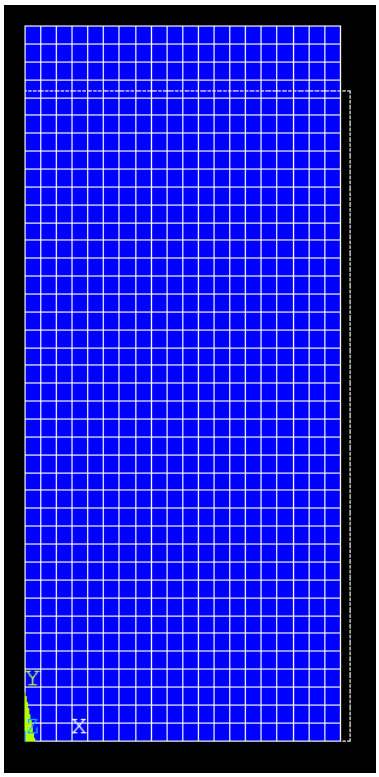


Figure 3.3: Deformed shape with magnification factor set to 10, front view.

Step 1.8: Save Model to .cdb File

Archive the model, writing the DB information to a .cdb file. From the **Preprocessor** menu, choose **Archive Model** and then **Write**. In the dialog box, switch Data to Archive to DB All finite element information and then set the file name to *rectangular_bar.cdb*. You can specify the correct folder by selecting the '**…**' button.

3.2 Step 2: Reading FE Model into FRANC3D

The next step is to insert a crack into the rectangular bar using FRANC3D. The steps are outlined below. We will compare the stress intensity factors based on far-field applied loading with those based on crack face tractions as computed from the uncracked stress field.

Step 2.1: Read the ANSYS Mesh

Start with the FRANC3D graphical user interface and select **File** and **Open**. Switch the File Filter in the Open Model File dialog box, Fig 3.4, to Ansys Files (*.cdb) and select the file name for the local model, called *rectangular_bar.cdb* here. Select **Accept**.

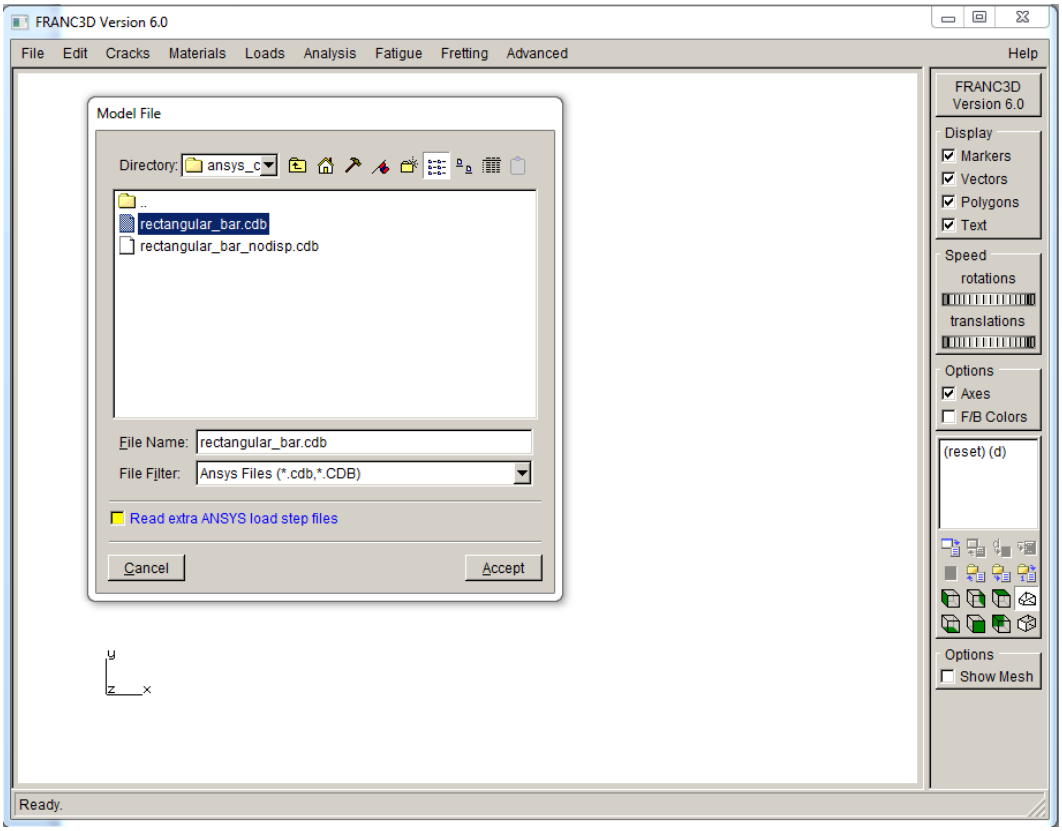


Figure 3.4: FRANC3D graphical user interface.

Step 2.2: Selecting the Retained Items in the Local FE Model

The dialog boxes shown in Fig 3.5 will be displayed, choose to retain selected items and select **Next**. Select all for boundary conditions and none for mesh facet groups, and then select **Finish**.

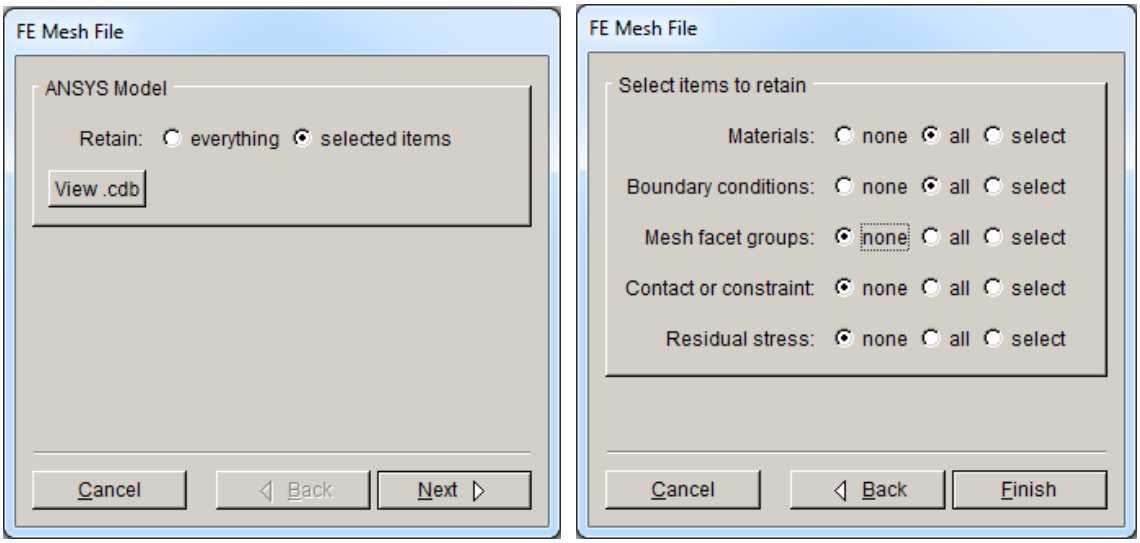


Figure 3.5: ANSYS Model retain dialog box.

The model will be displayed in the modeling window. You can turn on the surface mesh and manipulate the view. The model should appear as in Fig 3.6, which shows that the mesh is retained on the top and bottom surfaces.

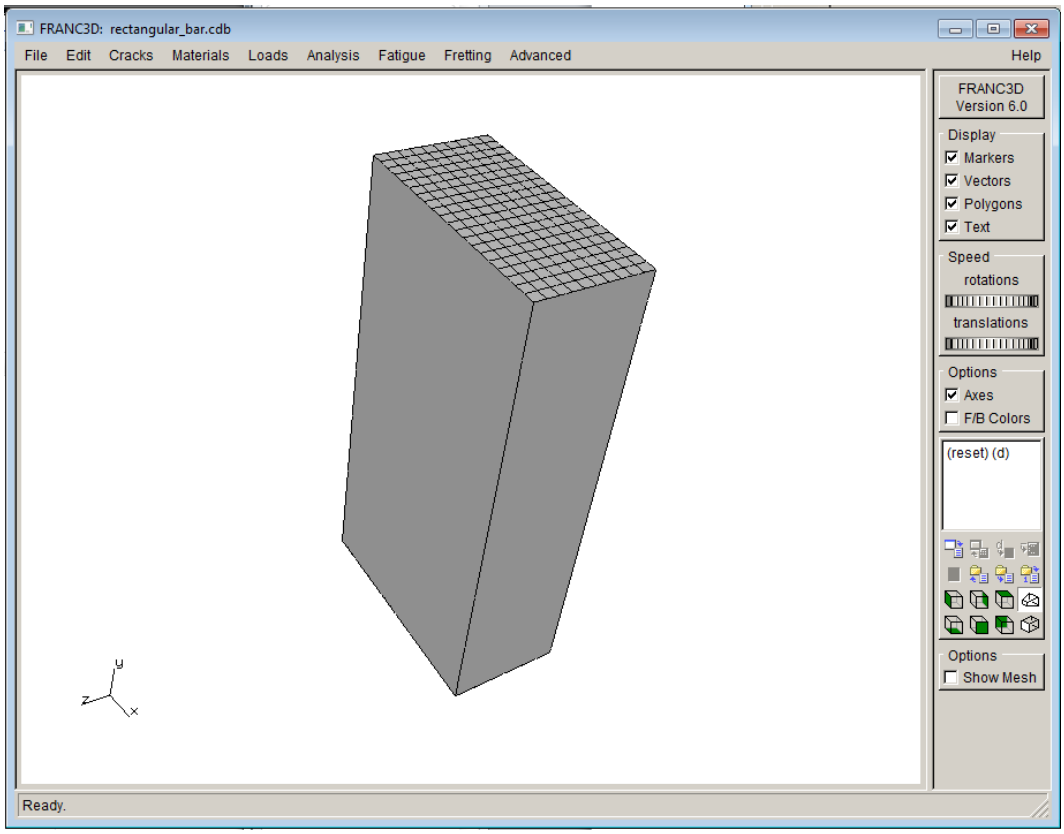


Figure 3.6: Local 3x3x3 ANSYS model converted to FRANC3D with top mesh faces retained.

3.3 Step 3: Insert a Crack

We will now insert a quarter-penny corner crack into the model.

Step 3.1: Select New Flaw Wizard and Flaw Type

From the FRANC3D menu, select **Cracks** and **New Flaw Wizard**. The first panel of the wizard should appear as in Fig 3.7. The default flaw type is Crack (zero volume flaw) and this is what we want, so select **Next**.

Step 3.2: Specify Crack Type

The next panel of the wizard, Fig 3.8, allows you to choose the type of crack. The default shape is the elliptical crack, which is what we want, so select **Next**.

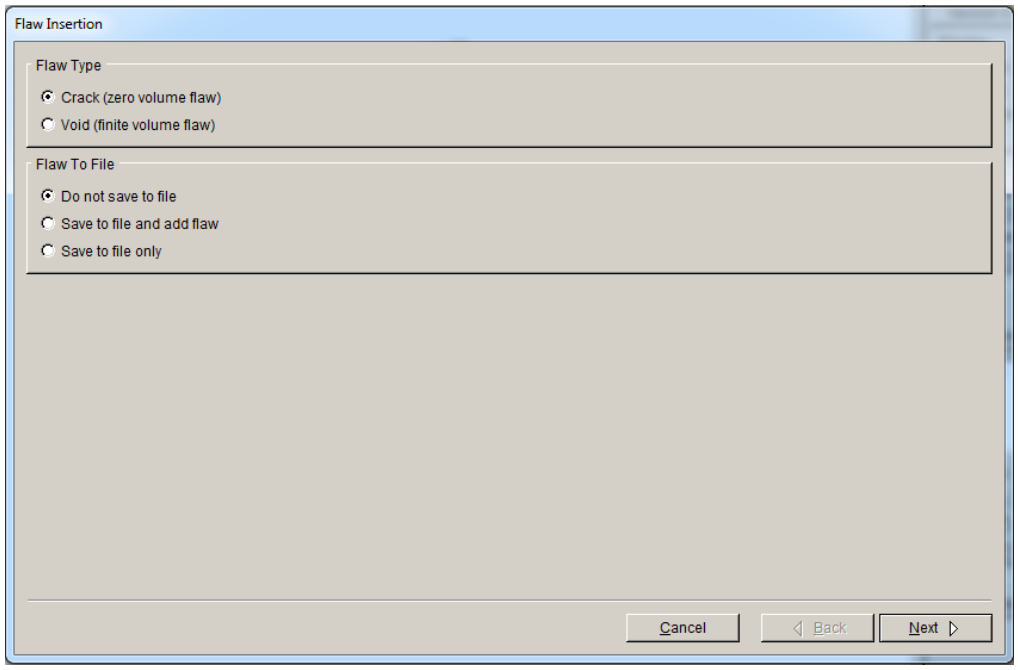


Figure 3.7: New flaw wizard first panel to choose flaw type.

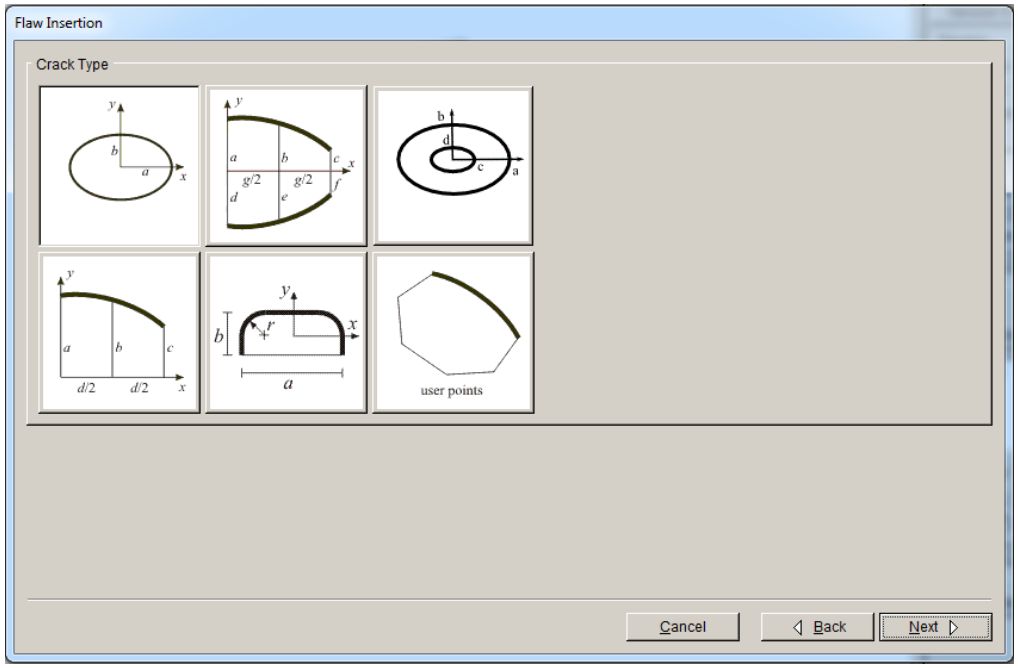


Figure 3.8: Flaw wizard panel to choose zero volume flaw type.

Step 3.3: Specify Crack Dimensions

The next panel of the wizard, Fig 3.9, allows you to specify the size of the ellipse. Enter 0.05 for both a and b and select **Next**.

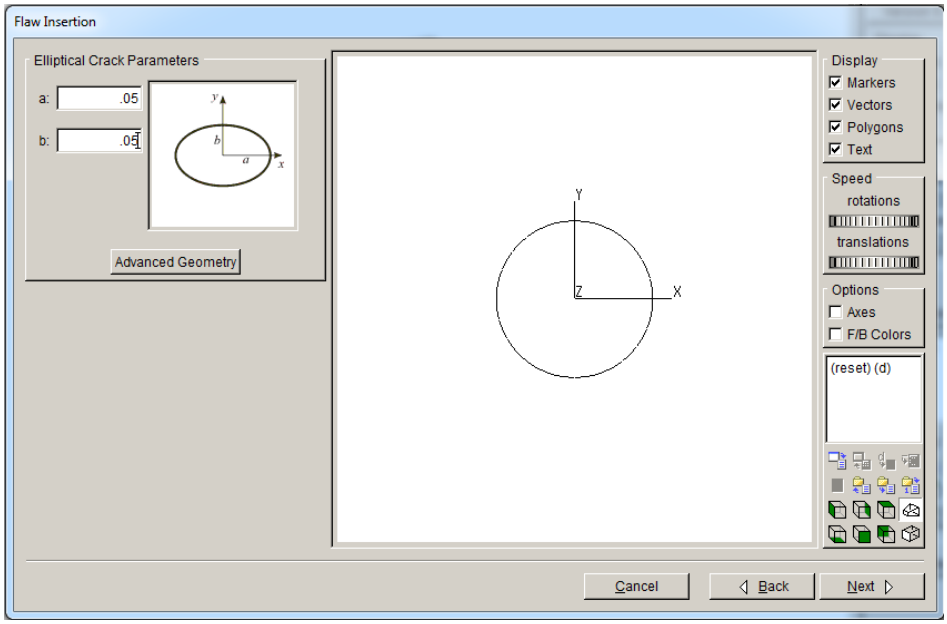


Figure 3.9: Flaw wizard panel to set size of ellipse.

Step 3.4: Specify Crack Location and Orientation

The next panel of the wizard, Fig 3.10, allows you to specify location and orientation of the flaw. Set the Translation for the X Axis to 0.5, for the Y Axis to 0.5, and for the Z axis to 0.25. Enter 90 for the 1st Rotation Angle and set the axis to X. Select **Next**.

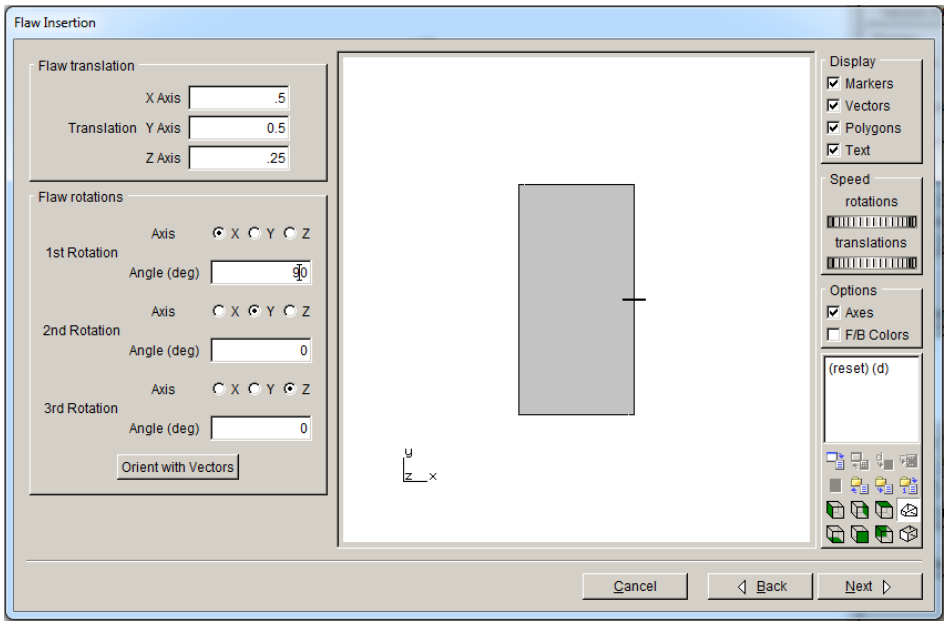


Figure 3.10: Flaw wizard panel to set location and orientation.

Step 3.5: Specify Crack Front Template

The next panel of the wizard, Fig 3.11, allows you to specify the crack front template parameters. We will leave all values at their defaults; select **Finish**.

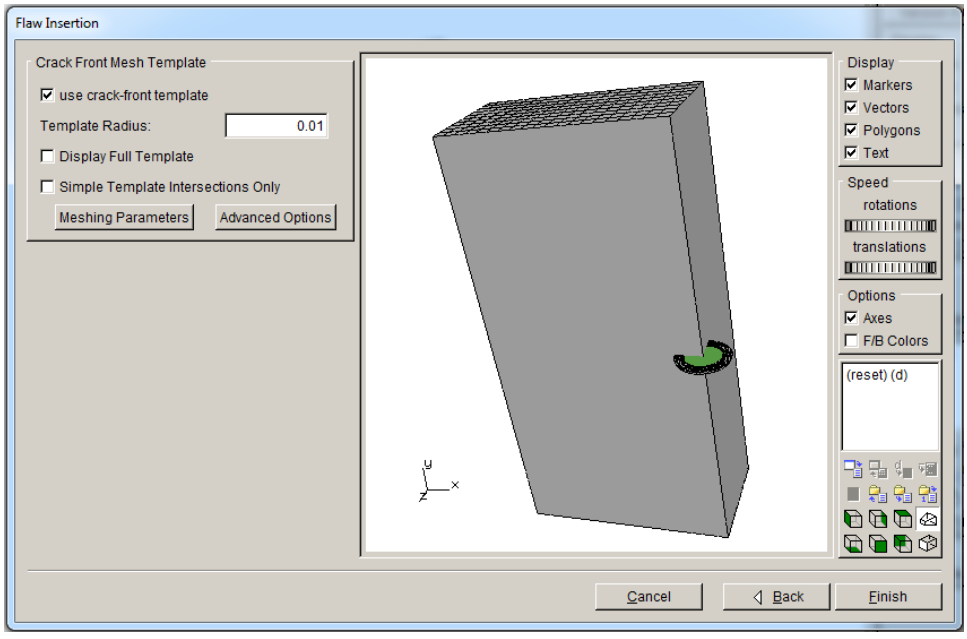


Figure 3.11: Flaw wizard panel to set crack front template parameters.

The program begins the process of inserting the flaw into the original model and then meshes the resulting cracked model. The progress of the operations is displayed on the screen, Fig 3.12. When meshing is complete, the Flaw Insertion Status box will disappear, and the re-meshed cracked model will be displayed, Fig 3.13.

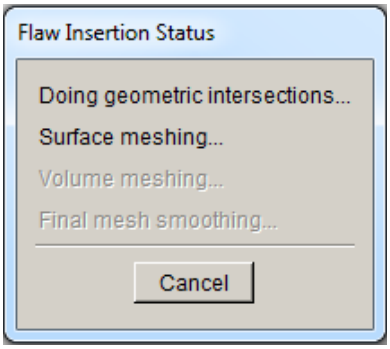


Figure 3.12: Flaw Insertion Status window.

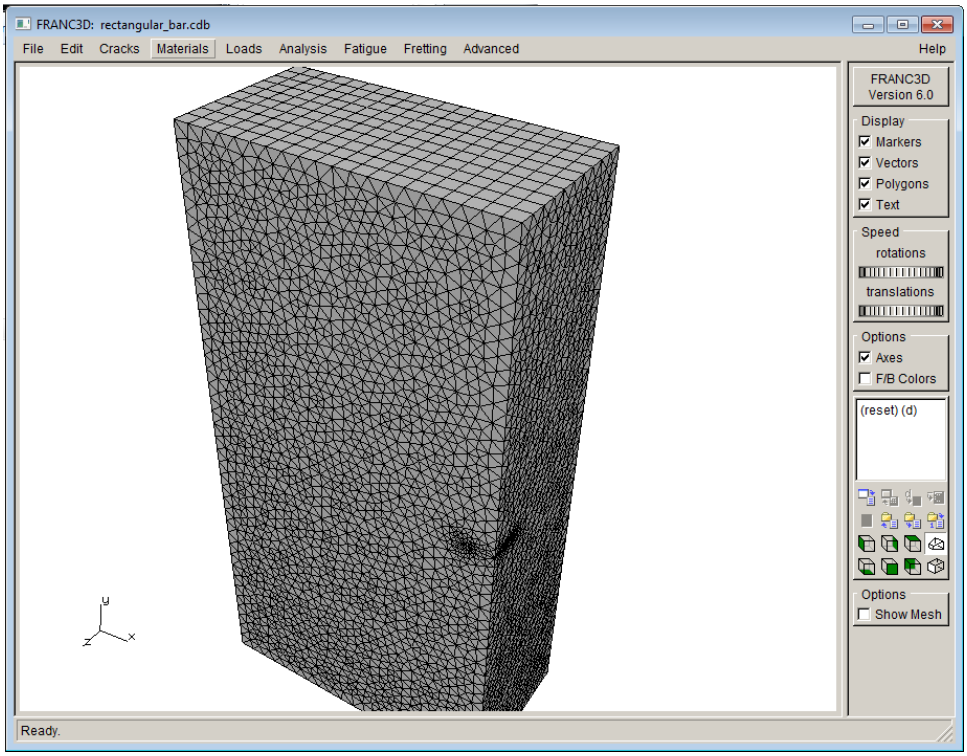


Figure 3.13: Meshed model with crack.

3.4 Step 4: Static Crack Analysis

We will now perform the stress analysis using ANSYS.

Step 4.1: Select Static Crack Analysis

From the FRANC3D menu, select **Analysis** and **Static Crack Analysis**. The first panel of the wizard should appear as in Fig 3.14. We will specify the file name for the FRANC3D database first. We call it *rectangular_bar_05crack.fdb*; select **Next** once you enter the File Name.

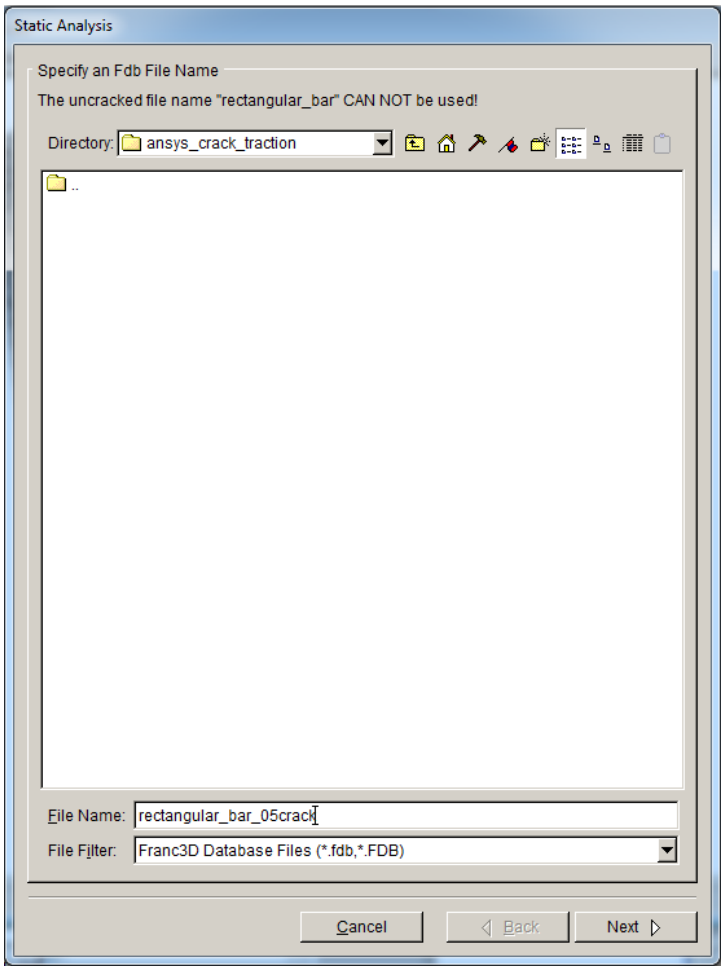


Figure 3.14: Static Analysis wizard first panel – File Name.

Step 4.2: Specify Analysis Code

The next panel of the wizard, Fig 3.15, allows us to specify the solver; choose ANSYS.

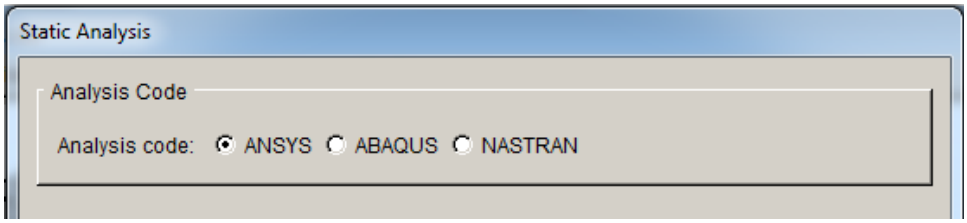


Figure 3.15: Static Analysis wizard second panel – solver.

Step 4.3: Specify Analysis Options

The next panel of the wizard, Fig 3.16, allows us to specify the ANSYS analysis and output options. We will not connect this model to a global model so we uncheck the Connect to global model box and then select **Finish.**

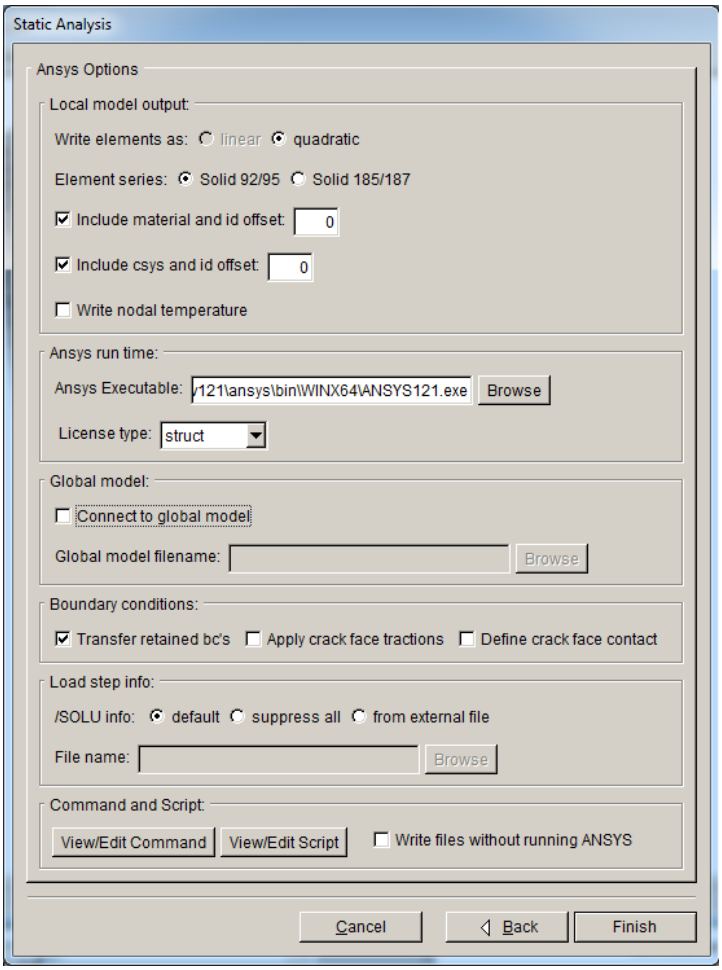

Static Analysis
Ansys Options
Local model output:
Write elements as: linear quadratic
Element series: Solid 92/95 Solid 185/187
Include material and id offset: 0
Include csys and id offset: 0
Write nodal temperature
Ansys run time:
Ansys Executable: v121\ansys\bin\WINX64\ANSYS121.exe Browse
License type: struct
Global model:
Connect to global model
Global model filename: Browse
Boundary conditions:
Transfer retained bc's Apply crack face tractions Define crack face contact
Load step info:
/SOLU info: default suppress all from external file
File name: Browse
Command and Script:
View/Edit Command View/Edit Script Write files without running ANSYS
Cancel Back Finish

Figure 3.16: Static Analysis wizard third panel – ANSYS output options.

ANSYS should start in the background. If ANSYS fails to start, the command line and macro files can be used to start the analysis outside of FRANC3D (from a cmd/terminal window).

3.5 Step 5: Compute Stress Intensity Factors

We will now compute the stress intensity factors for this crack. If ANSYS ran successfully from FRANC3D, then the model should be open and the displacement file will be read automatically and you can skip to Step 5.2.

Step 5.1: Restart FRANC3D

From the FRANC3D menu, select **File** and **Open**. Choose the *rectangular_bar_05crack.fdb* file and select **Accept**.

Note: you might want to close the previous model or restart FRANC3D.

Step 5.2: Select Compute SIFs

From the FRANC3D menu, select **Cracks** and **Compute SIFs**. The Stress Intensity Factor wizard is displayed, Fig 3.17. You should use the M-Integral, but you can check that the Displacement Correlation results are similar. There are no thermal or crack face traction terms. When you select **Finish**, the SIFs Plot dialog is displayed, Fig 3.18. You can view the three stress intensity factor modes and export the data.

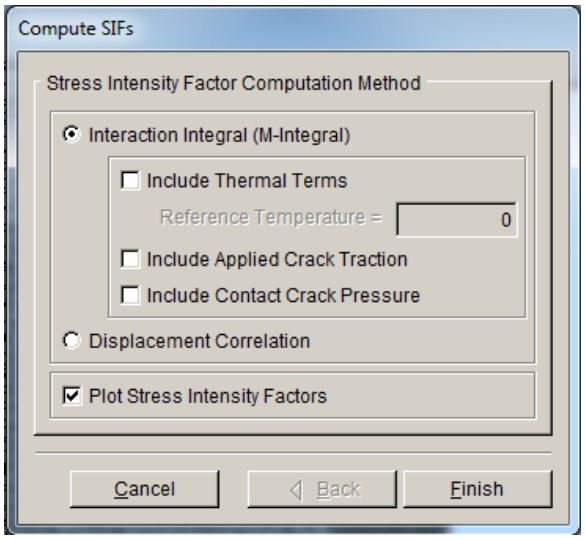


Figure 3.17: Compute SIFs panel.

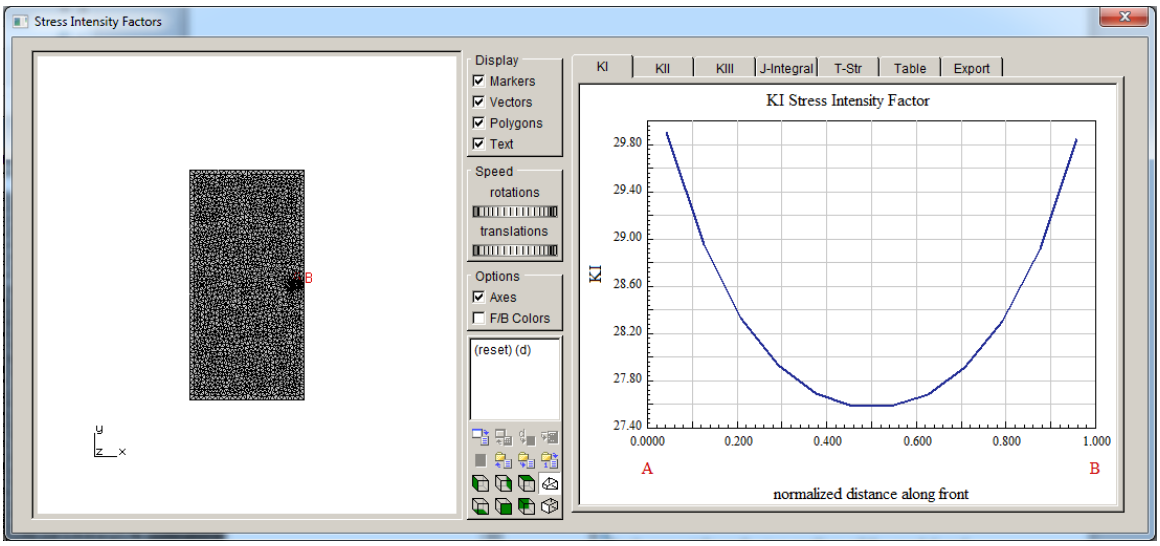


Figure 3.18: Stress Intensity Factor dialog.

3.6 Step 6: Crack Analysis with Crack Face Traction

This step of the tutorial describes how to apply crack face tractions using the external mesh and stress from the uncracked analysis. The first step is to make sure that we have saved the nodal stress listing for the rectangular bar with applied displacement. Return to ANSYS if needed; make sure that you have turned off **PowerGraphics** before listing the nodal stress components (see Step 1.7 above). The second step is to remove the applied y-displacement from the top surface of the model and resave the *.cdb* file, calling it *rectangular_bar_nodisp.cdb*.

Step 6.1: Read the FE Model

Repeat Step 2 in Section 3.2, but at Step 2.1, read the *rectangular_bar_nodisp.cdb* file. Choose to retain selected items. We choose to retain all materials and all of the boundary conditions, which now should only be the constraints on the bottom surface.

Step 6.2: Insert a Crack

Repeat Step 3 in Section 3.3. Once we have inserted the crack and gotten a mesh, we can proceed with Step 6.3 listed below.

Step 6.3: Select Crack Face Pressure/Traction

From the FRANC3D menu, select **Loads** and **Crack Face Pressure/Traction**. The first panel of the wizard should appear as in Fig 3.19. Select **Add** to define a new entry, Fig 3.20.

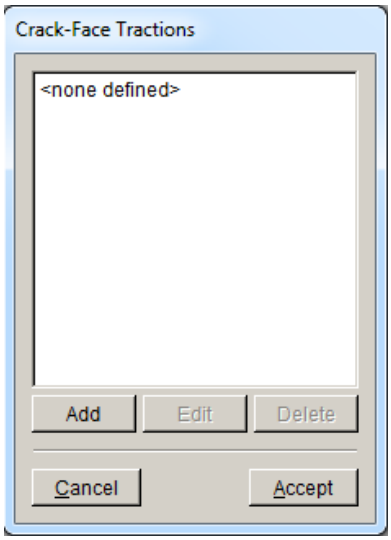


Figure 3.19: Crack Face Tractions dialog.

Step 6.4: Specify Residual Stress Defined on Mesh

In the dialog box shown in Fig 3.20, choose Residual Stress Defined on a Mesh, and select **Next**. In the next dialog box, Fig 3.21, we provide the file name for the mesh and the stress. We saved the *cdb* and *str* files previously for the uncracked bar; this is where we use these

files. The next panel, Fig 3.22 lists the available load steps in the results file. Select **Finish** to return to the list of crack face tractions; there should be one entry in the list now. Select **Accept** from the dialog shown in Fig 3.19.

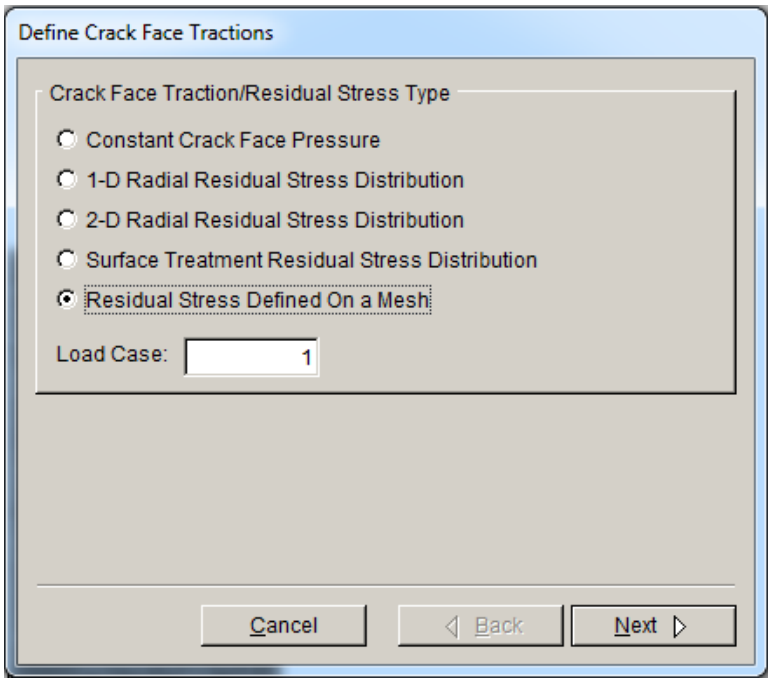


Figure 3.20: Define Crack Face Tractions dialog.

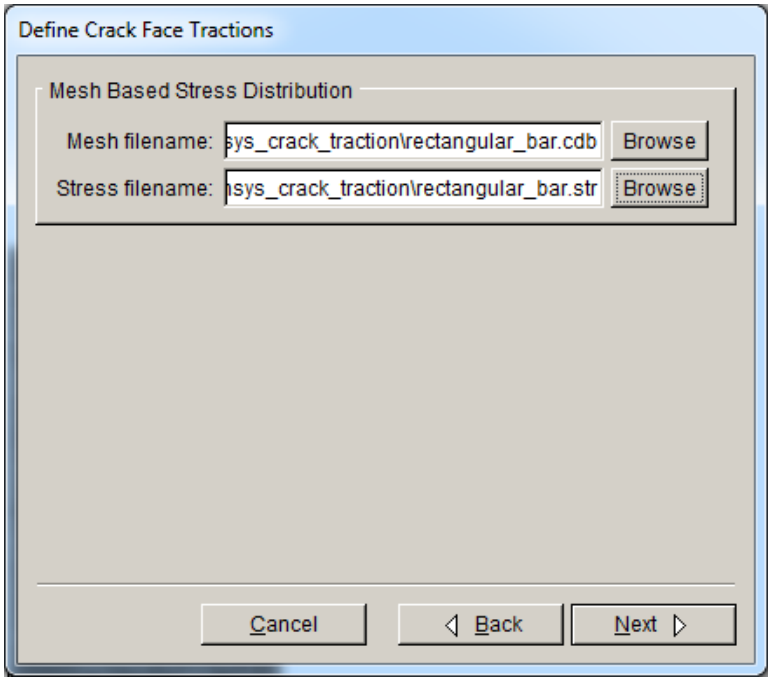


Figure 3.21: Define Crack Face Tractions: Mesh Based Stress Distribution dialog.

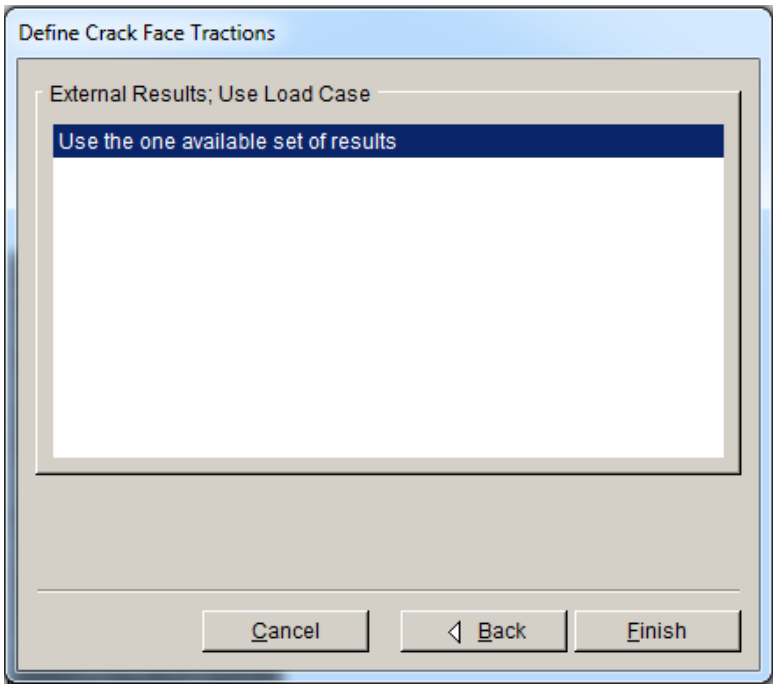


Figure 3.22: Define Crack Face Tractions: Available load steps in results file.

Step 6.5: Static Crack Analysis

Repeat Step 4 in Section 3.4. However, we will choose to Apply crack face tractions, Fig 3.23, in addition to transferring all retained boundary conditions. We also provide a different file name: *rectangular_bar_05crack_withtract.fdb*.

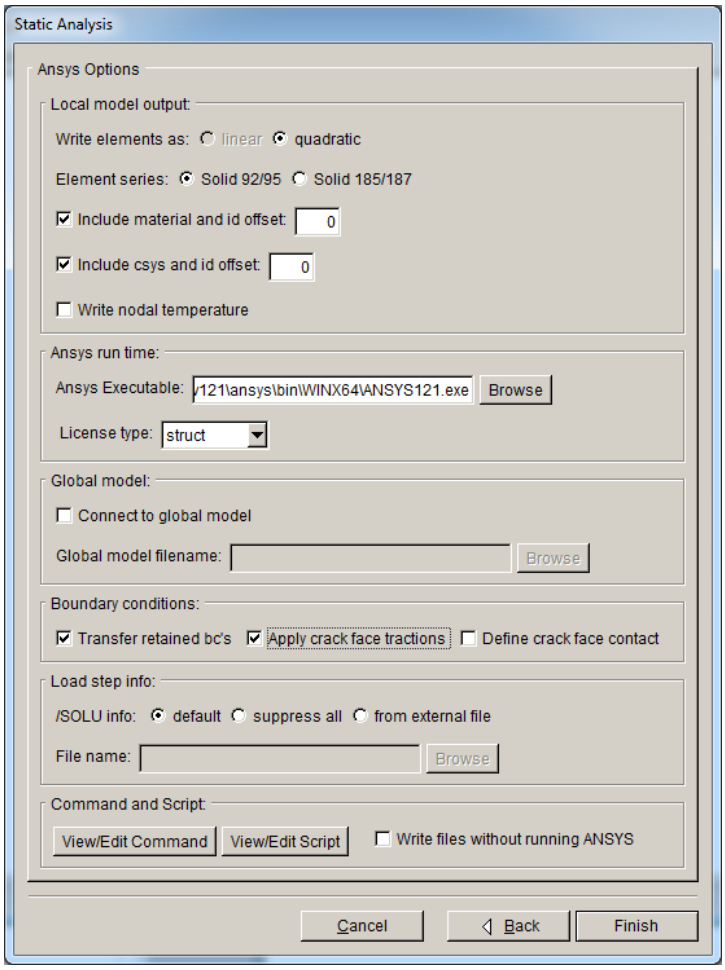

Static Analysis
Ansys Options
Local model output:
Write elements as: linear quadratic
Element series: Solid 92/95 Solid 185/187
Include material and id offset: 0
Include csys and id offset: 0
Write nodal temperature
Ansys run time:
Ansys Executable: v121\ansys\bin\WINX64\ANSYS121.exe Browse
License type: struct
Global model:
Connect to global model
Global model filename: Browse
Boundary conditions:
Transfer retained bc's Apply crack face tractions Define crack face contact
Load step info:
/SOLU info: default suppress all from external file
File name: Browse
Command and Script:
View/Edit Command View/Edit Script Write files without running ANSYS
Cancel Back Finish

Figure 3.22: Static analysis panel – crack face tractions applied.

Step 6.6: Compute Stress Intensity Factors

We will now compute the stress intensity factors for this crack, following Step 5 in Section 3.5. In this case, we have crack face traction terms so this box should be checked when using the M-integral, Fig 3.23. The mode I stress intensity factors are shown in Fig 3.24. You can compute the SIFs with the applied crack traction unchecked for comparison. You can also check the SIFs using displacement correlation.

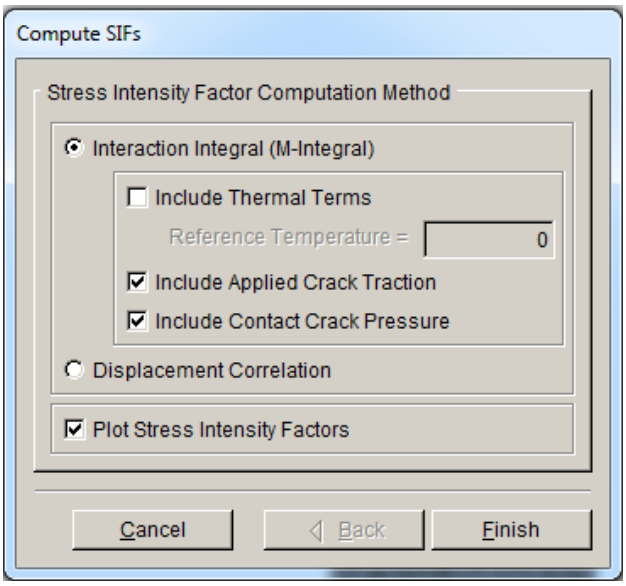


Figure 3.23: Stress Intensity Factor Computation Method dialog.

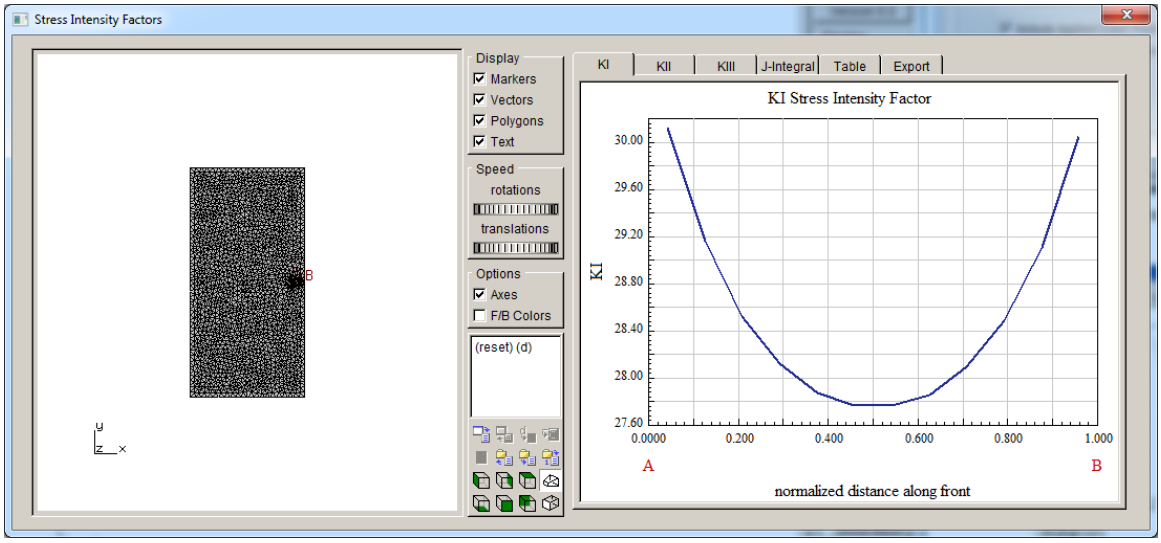


Figure 3.24: Stress Intensity Factor dialog showing the Mode I SIF.

3.7 Step 7: Comparison of Stress Intensity Factors

It is noted that the Mode I SIF values for far-field loading are slightly different than those for crack face tractions, Fig 3.25. The values when using crack face tractions are about 0.7% higher. The results could likely be improved by refining the mesh at the crack front. We can do this by

starting over with the uncracked *.cdb* file and reinserting the flaw, but choosing a smaller template radius (see Fig 3.11); this is left as an exercise for the reader.

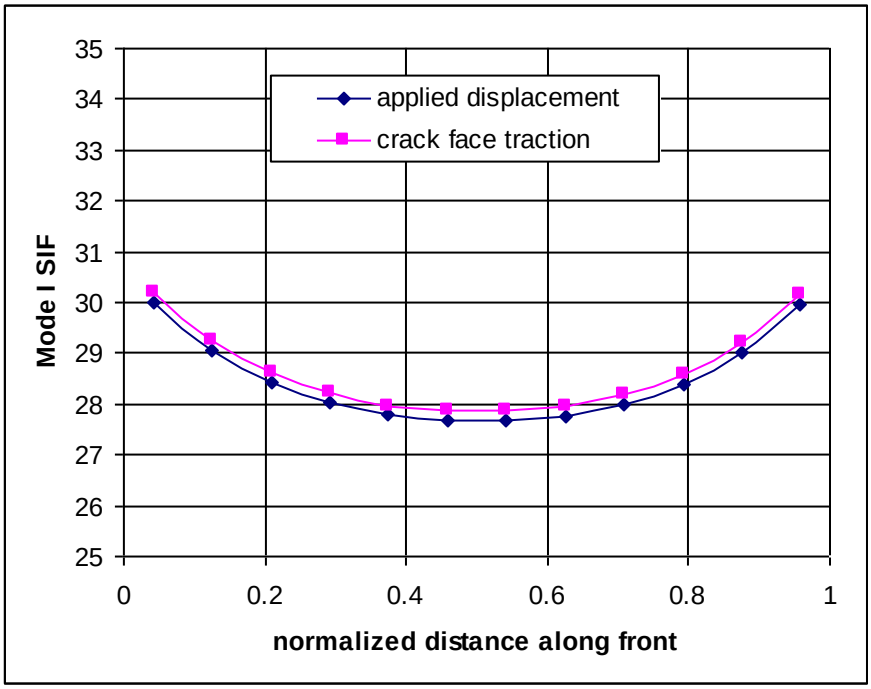


Figure 3.25: Stress Intensity Factor dialog showing the Mode I SIF.

4.0 Tutorial 3: Center Through-Crack in a Plate Sub-Domain, with Two Crack Fronts, Two Load Cases, and a SIF History

In this tutorial, we describe the steps to complete an automated crack growth analysis using the FRANC3D and ANSYS interface; this will include the growth of multiple crack fronts and the analysis of multiple load cases. For this tutorial, an initial uncracked model will be created and analyzed in ANSYS. The tutorial is divided into 6 major steps:

1. Creating the uncracked geometry and mesh using ANSYS;
2. Crack insertion and static analysis;
3. Crack growth of two fronts;
4. Automated crack growth;
5. SIF history extraction; and
6. Analysis with multiple load cases.

4.1 Step 1: Create the uncracked model using ANSYS

Start by creating a simple plate model using ANSYS. The plate dimensions are x=20, y=50 and z=5 starting from the global Cartesian origin. The chosen element type is a solid 20-noded brick. The material properties are linear elastic with E=10,000 and Poisson's ratio=0.3. The boundary conditions consist of unit traction equal to 1.0 on the upper y-surface and y-constraint on the lower surface. The origin is pinned and the node at (0,0,5) is constrained in the x-direction also. The mesh consists of 10 elements along the x-axis, 25 elements along the y-axis and 3 elements along the z-axis, Fig 4.1 – left panel. We extract the middle portion of the plate and define cut_surface node components. The global portion, right panel of Fig 4.1, and the local portion are saved to *.cdb* files.

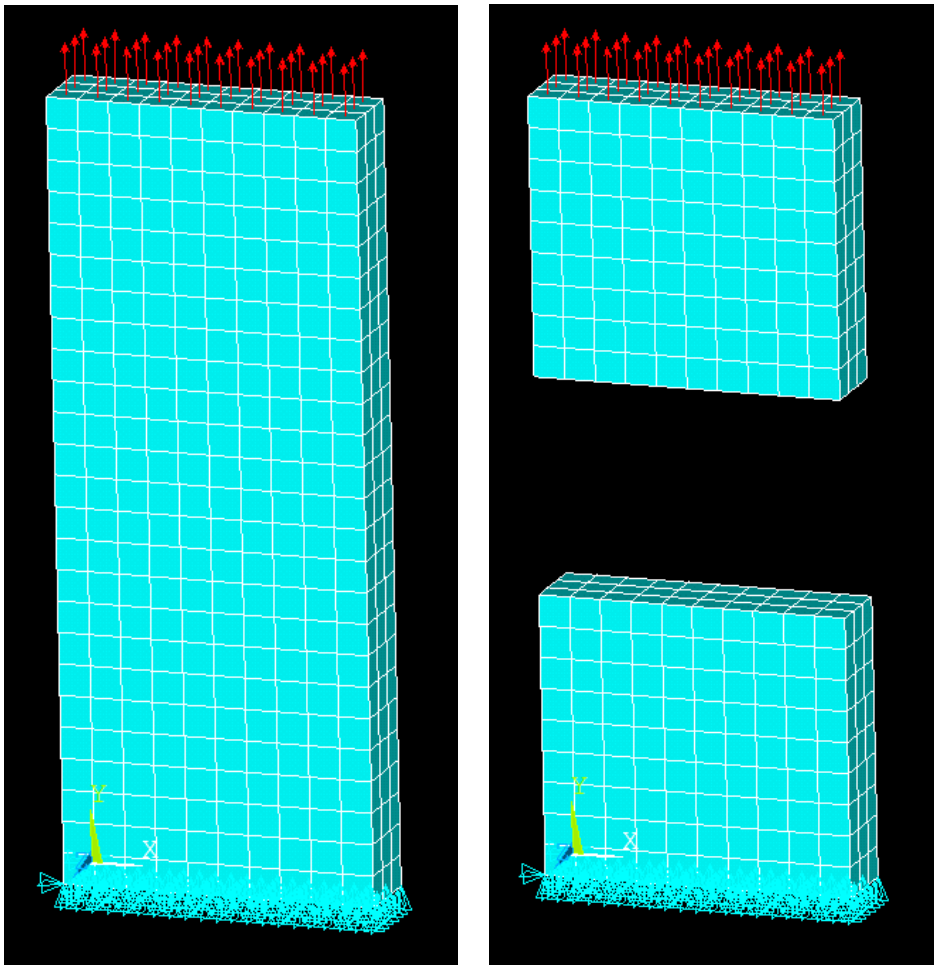


Figure 4.1: ANSYS full plate model (left panel) and 'global' model (right panel) with boundary conditions displayed.

4.2 Step 2: Crack Insertion and Static Analysis

The next step is to insert a crack into the plate using FRANC3D and then perform the static analysis using ANSYS. The steps are outlined below.

Step 2.1: Read the FE Mesh and Select Retained Data

Start with the FRANC3D graphical user interface and select **File** and **Open**. Switch the File Filter in the Open Model File dialog box to Ansys Files (*.cdb) and select the file name for the local model, called *plate_local.cdb* here. Select **Accept**.

Choose to retain selected items and then choose to select Mesh facet groups. We will retain the CUT_SURF node component that defines the cut surface between the local and global ANSYS model portions. The model should appear as in Fig 4.2, with the mesh facets retained on the upper and lower cut surfaces.

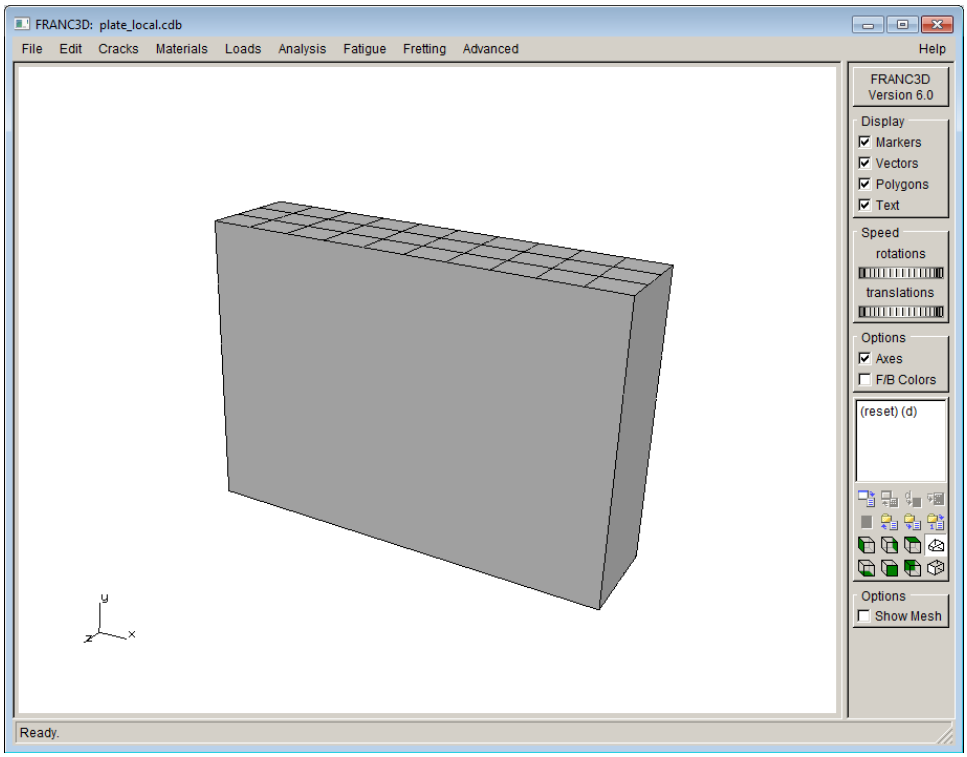


Figure 4.2: Local plate model with retained mesh facets on upper surface.

Step 2.2: Crack Insertion

From the FRANC3D menu, select **Cracks** and **New Flaw Wizard**. The default flaw type is Crack (zero volume flaw) and this is what we want. We will insert a through-crack into the plate, so choose that crack type and then set the crack dimensions based on Fig 4.3 and the crack location/orientation based on Fig 4.4. The crack front template mesh defaults are okay, so you can proceed with inserting the crack geometry into the local plate model. The remeshed cracked local portion should appear as in Fig 4.5.

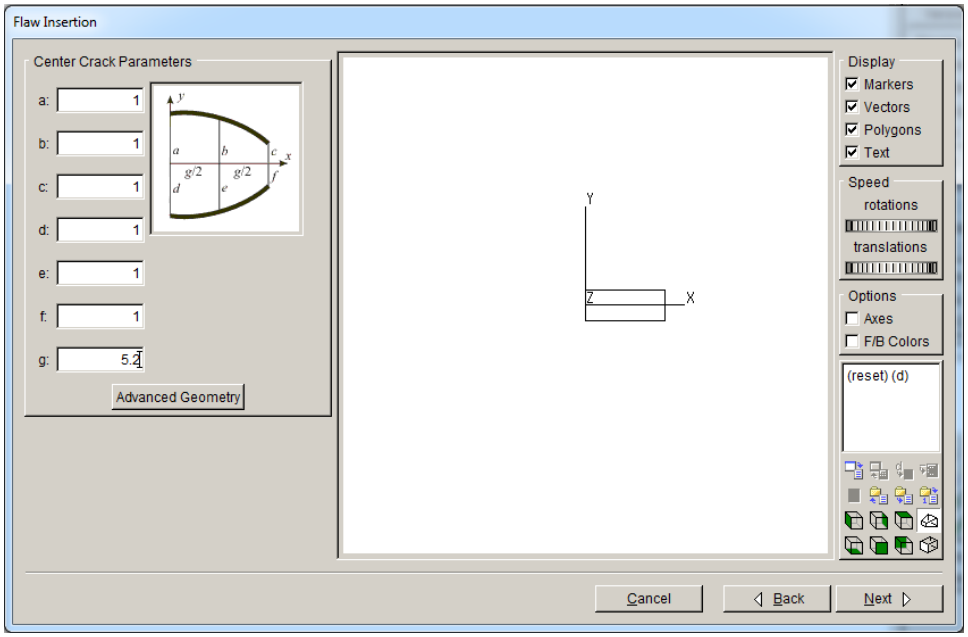


Figure 4.3: Through-crack dimensions for plate model.

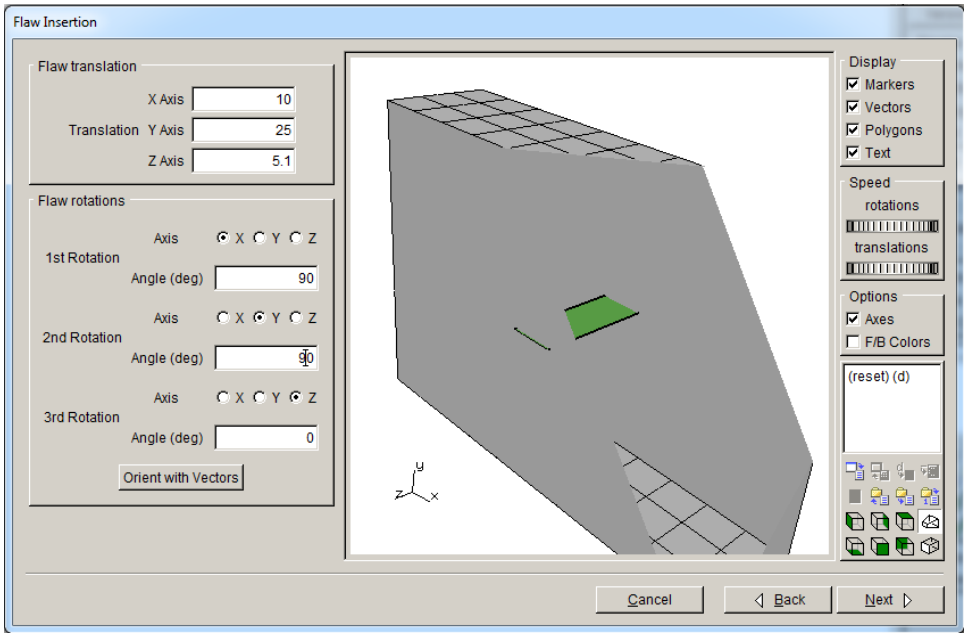


Figure 4.4: Through-crack location and orientation for plate model.

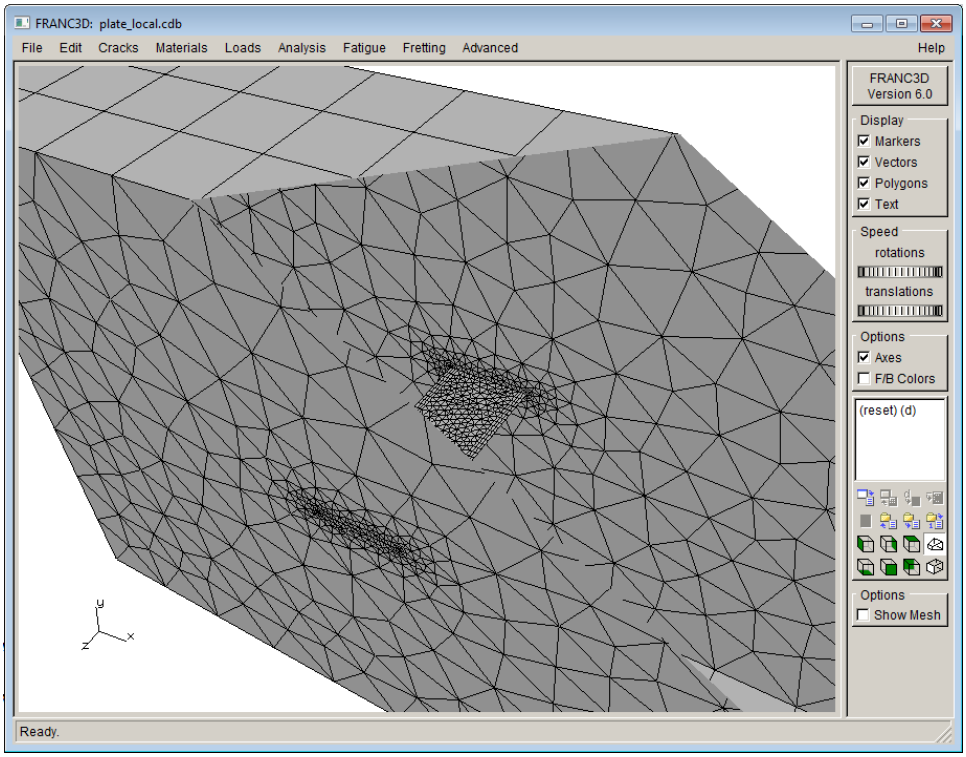


Figure 4.5: Meshed through-crack in the local portion of the plate model.

Step 2.3: Static Crack Analysis

From the FRANC3D menu, select **Analysis** and **Static Crack Analysis**. We specify the file name: *plate_crack.fdb*. We choose ANSYS as the solver and then set the global model to connect with as *plate_global.cdb*, Fig 4.6 – left panel, and then specify the CUT_SURF node component for merging the model portions together, Fig 4.6 – right panel.

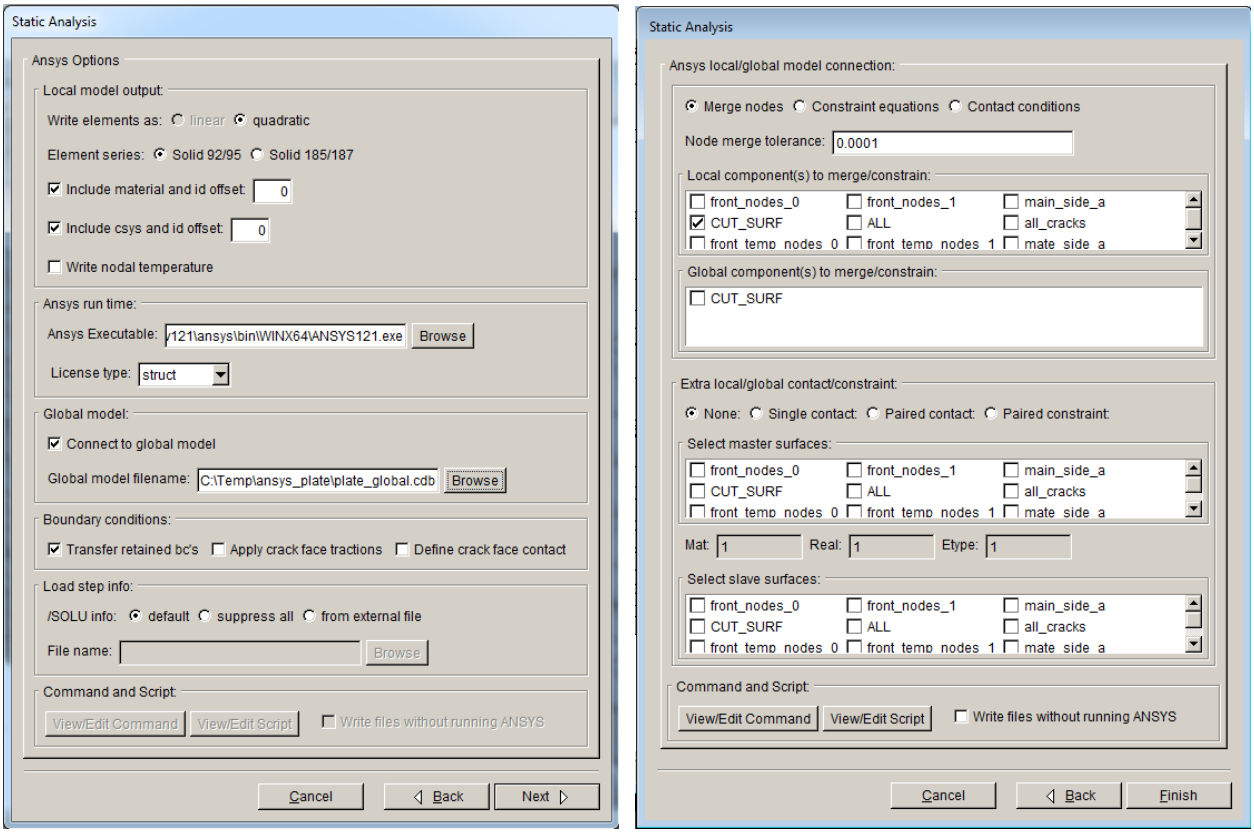

Static Analysis
Ansys Options
Local model output:
Write elements as: linear quadratic
Element series: Solid 92/95 Solid 185/187
Include material and id offset: 0
Include csys and id offset: 0
Write nodal temperature
Ansys run time:
Ansys Executable: v121\ansys\bin\WINX64\ANSYS121.exe Browse
License type: struct
Global model:
Connect to global model
Global model filename: C:\Temp\ansys_plate\plate_global.cdb Browse
Boundary conditions:
Transfer retained bc's Apply crack face tractions Define crack face contact
Load step info:
/SOLU info: default suppress all from external file
File name: Browse
Command and Script:
View/Edit Command View/Edit Script Write files without running ANSYS
Cancel Back Next
Static Analysis
Ansys local/global model connection:
Merge nodes Constraint equations Contact conditions
Node merge tolerance: 0.0001
Local component(s) to merge/constrain:
front_nodes_0 front_nodes_1 main_side_a
CUT_SURF ALL all_cracks
front_temp_nodes_0 front_temp_nodes_1 mate_side_a
Global component(s) to merge/constrain:
CUT_SURF
Extra local/global contact/constraint:
None: Single contact: Paired contact: Paired constraint:
Select master surfaces:
front_nodes_0 front_nodes_1 main_side_a
CUT_SURF ALL all_cracks
front_temp_nodes_0 front_temp_nodes_1 mate_side_a
Mat: 1 Real: 1 Etype: 1
Select slave surfaces:
front_nodes_0 front_nodes_1 main_side_a
CUT_SURF ALL all_cracks
front_temp_nodes_0 front_temp_nodes_1 mate_side_a
Command and Script:
View/Edit Command View/Edit Script Write files without running ANSYS
Cancel Back Finish

Figure 4.6: Meshed through-crack in the local portion of the plate model.

4.3 Step 3: Compute SIFs and Grow Crack Fronts

We will now compute the stress intensity factors for this crack and then propagate the two crack fronts.

Step 3.1: Compute SIFs

From the FRANC3D menu, select **Cracks** and **Compute SIFs**. There will be two display windows showing the SIFs for the two crack fronts. The SIFs should basically be identical as shown in Fig 4.7.

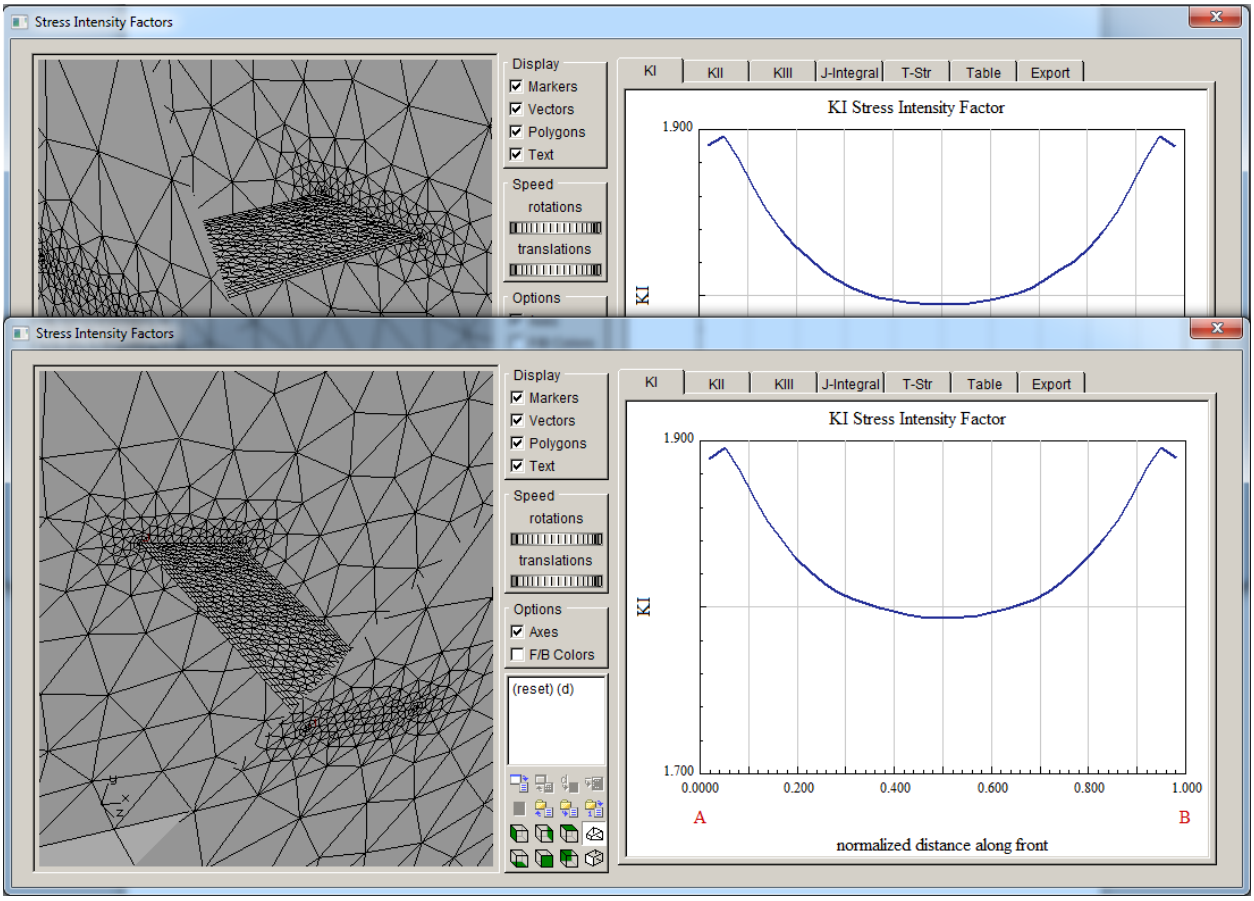


Figure 4.7: Mode I SIFs for both crack fronts of the through-crack in the plate model.

Step 3.2: Crack Growth of Two Crack Fronts

From the FRANC3D menu, select **Cracks** and **Grow Crack**. We can choose Quasi-Static crack growth and leave all the other options and values at their defaults. The two crack fronts grow based on a single crack growth rule and the user-specified median extension. The front fitting options can be set independently for each front, Fig 4.8. In this model, the crack growth should be the same for both fronts. You can proceed with the process of defining the crack front template mesh and remeshing the propagated crack.

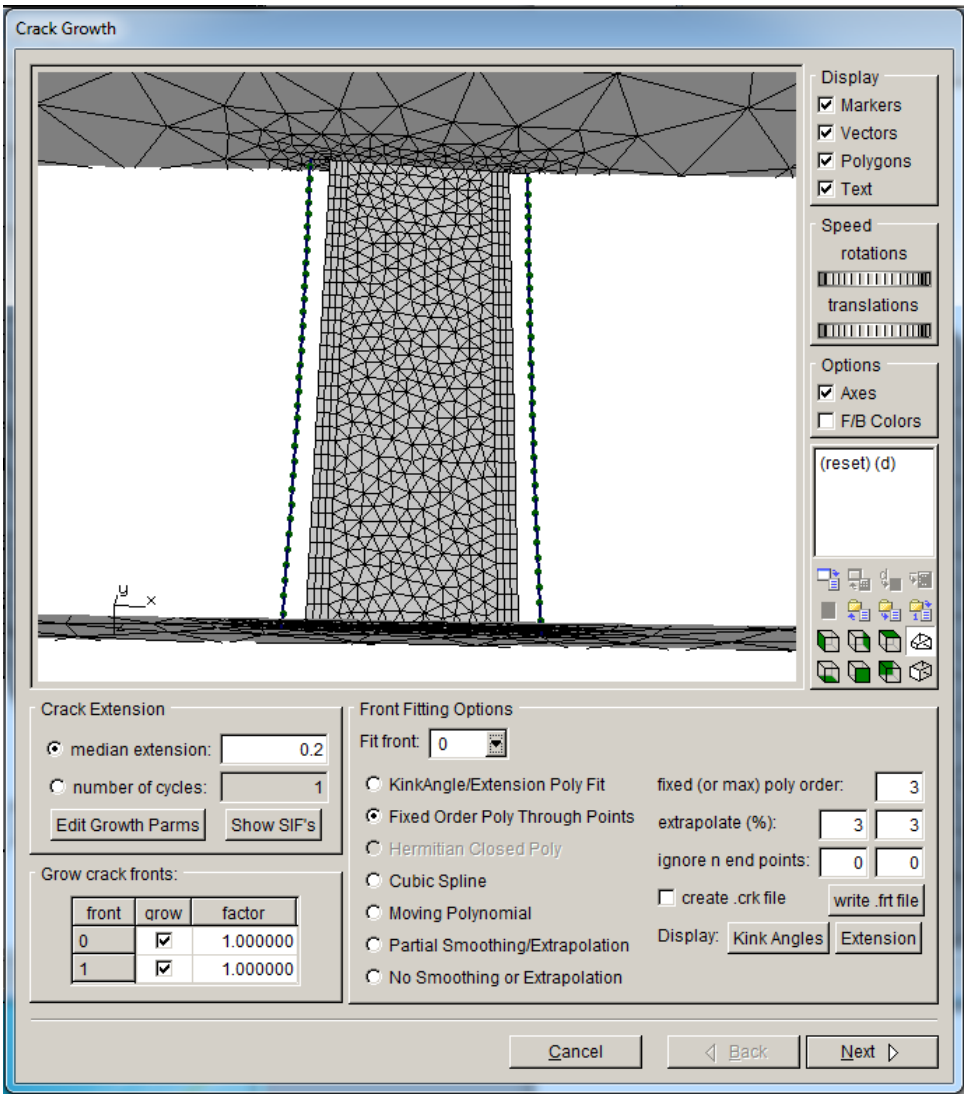


Figure 4.8: Two crack fronts propagate, with their own front fitting options.

4.4 Step 4: Automatic Crack Growth Analysis

In this step, we will perform automated crack growth analyses.

Step 4.1: Run Automatic Crack Growth Analysis

This step assumes that you successfully propagated both crack fronts in Section 4.2. From the FRANC3D menu, select **Analysis** and **Crack Growth Analysis**. We choose Quasi-Static crack growth again. We set the number of crack growth steps to 5 and use a constant

median extension of 0.2. We set the extrapolation for front fitting to 5% for both ends of both fronts. We set the base name as *plate_crack* and leave the current crack growth step id as 1. We connect to the global model: *plate_global.cdb* as in Section 4.2 and choose to merge the nodes based on the CUT_SURF component.

Step 4.2: Monitor Analyses

You can monitor the progress of the analyses; there should be a set of FRANC3D and ANSYS files for *_STEP_001* through *_STEP_006*.

4.5 Step 5: SIF History

In this step, we illustrate the process of extracting SIF history data using the 5 steps of crack growth completed in Section 4.4.

Step 5.1: Create Growth History

Continuing in FRANC3D with the model from Step 4, select **Advanced** and **Create Growth History**. The dialog shown in the left panel of Fig 4.9 will appear. You can use the **Plot** menu to display the crack fronts, Fig 4.9 – right panel. Close the dialog boxes when you have examined them; consult the FRANC3D Reference documentation for more details.

Step 5.2: Extract SIF History

From the FRANC3D main menu, select **Fatigue** and then **SIF History** to display the SIF History dialog, Fig 4.10. This dialog allows you to select a path through the crack fronts, plot the SIF along this path, and export this data for use in the Fatigue Life module. If you have multiple crack fronts, you can choose the starting crack front id through the **Settings** menu, which brings up the dialog shown in Fig 4.11.

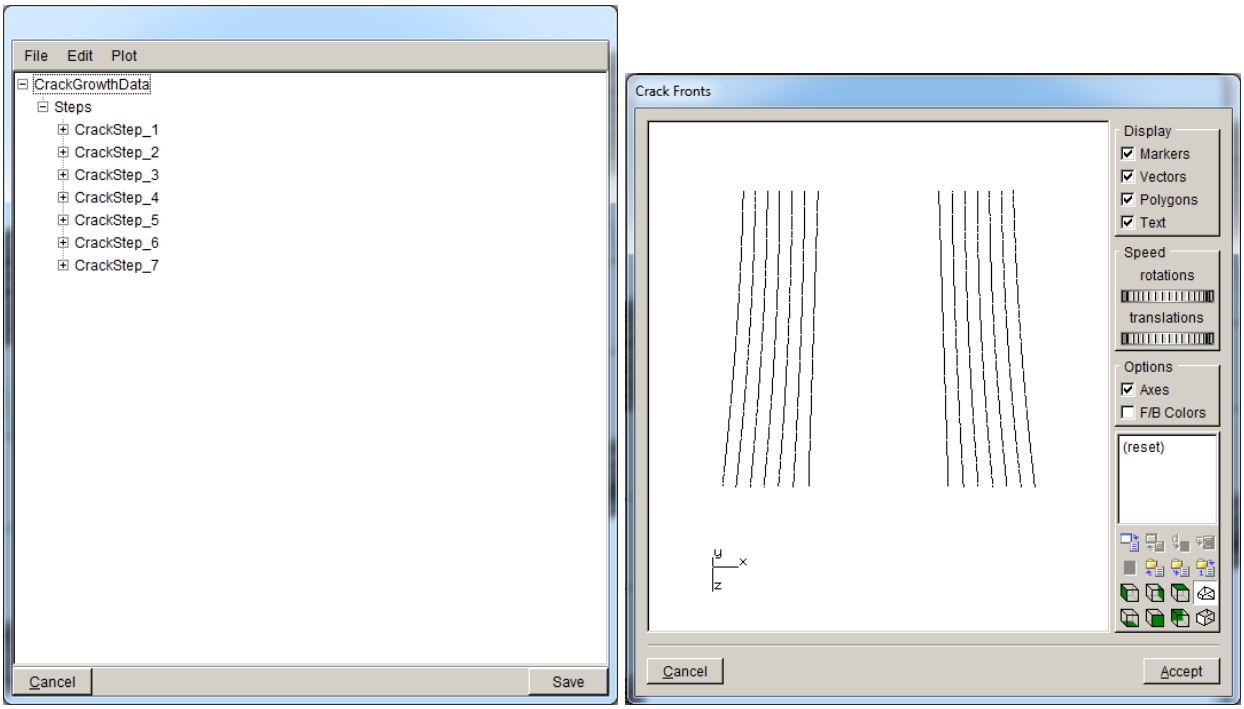


Figure 4.9: Create Growth History dialog and the six (x2) crack fronts displayed in the right panel.

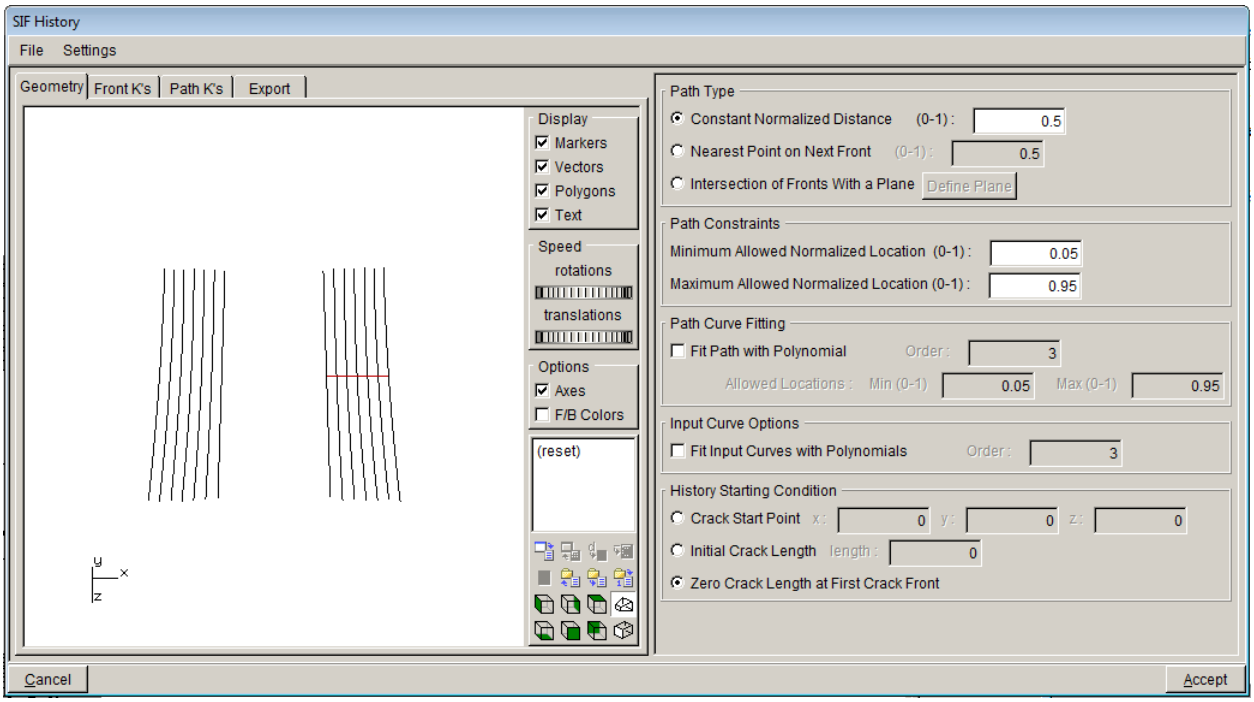


Figure 4.10: SIF History dialog.

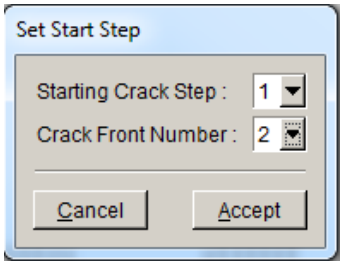


Figure 4.11: SIF history crack front start step dialog.

4.6 Step 6: Multiple Load Cases

In this step, we illustrate the analysis of multiple load cases. We will start from the initial crack configuration from Section 4.2.

Step 6.1: *Restart FRANC3D*

Start with the FRANC3D graphical user interface and select **File** and **Open**. Find the file: *plate_crack.fdb* and select **Accept**.

Step 6.2: Apply Crack Face Pressure/Traction

Select **Loads** and **Crack Face Pressure/Traction**. Click on **Add** in the dialog box shown in the left panel of Fig 4.12. Choose Constant Crack Face Pressure, which is the default, set the Load Case number to 2, and click on **Next** in the second dialog shown in the right panel of Fig 4.12. Set the Pressure value to 1 in the next dialog and select **Finish**. Note that a positive value of pressure will tend to open the crack. Finally click **Accept** on the original dialog.

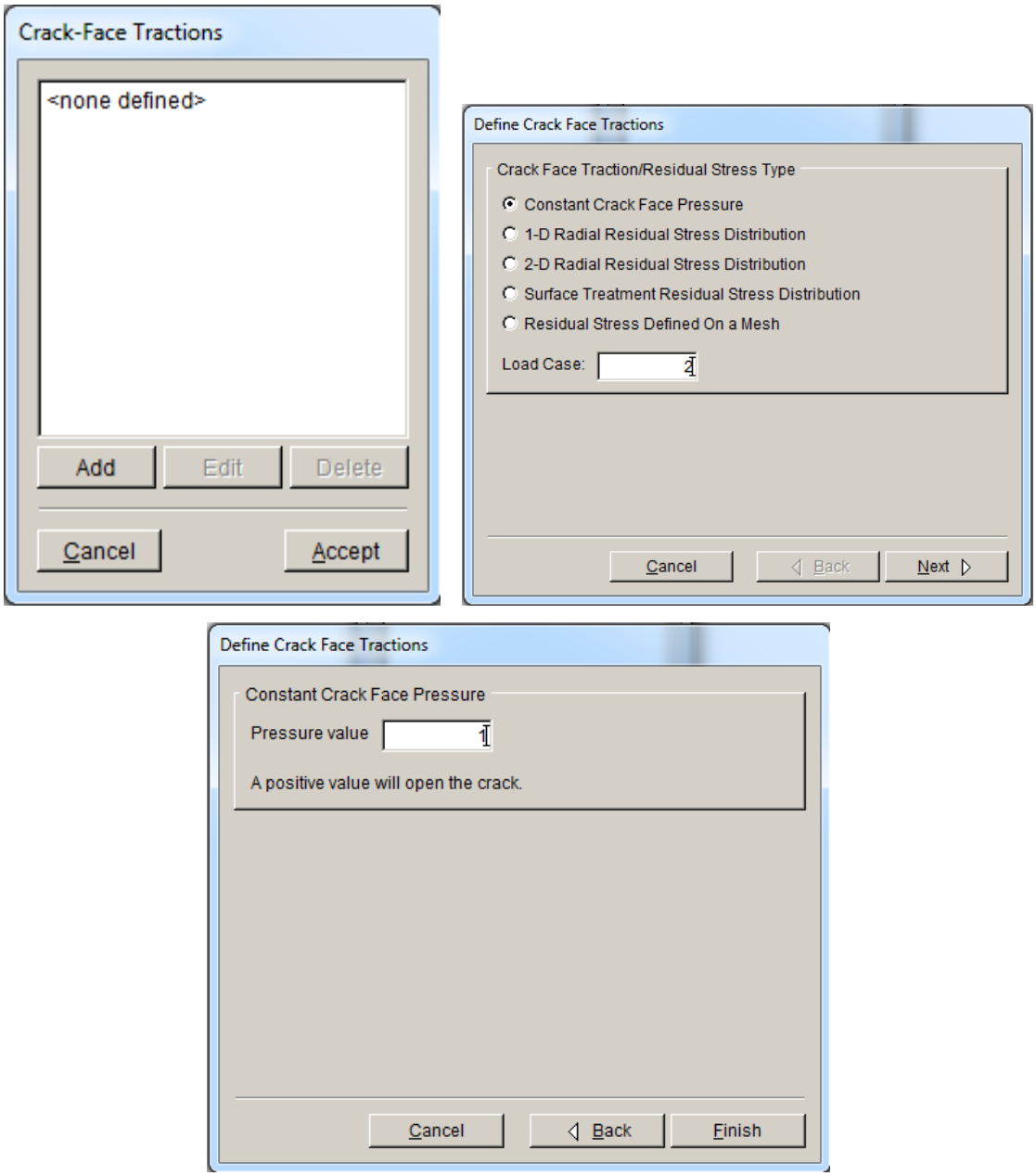


Figure 4.12 Crack face tractions dialog.

Step 6.3: Run Static Analysis

Perform a static crack analysis on this model to compute the SIFs. Analyze the model using ANSYS. The elements should be second order. All boundary conditions should be transferred and we must check the apply crack face tractions box, Fig 4.13. We connect to the *global_plate.cdb* model and select the CUT_SURF node component for merging nodes.

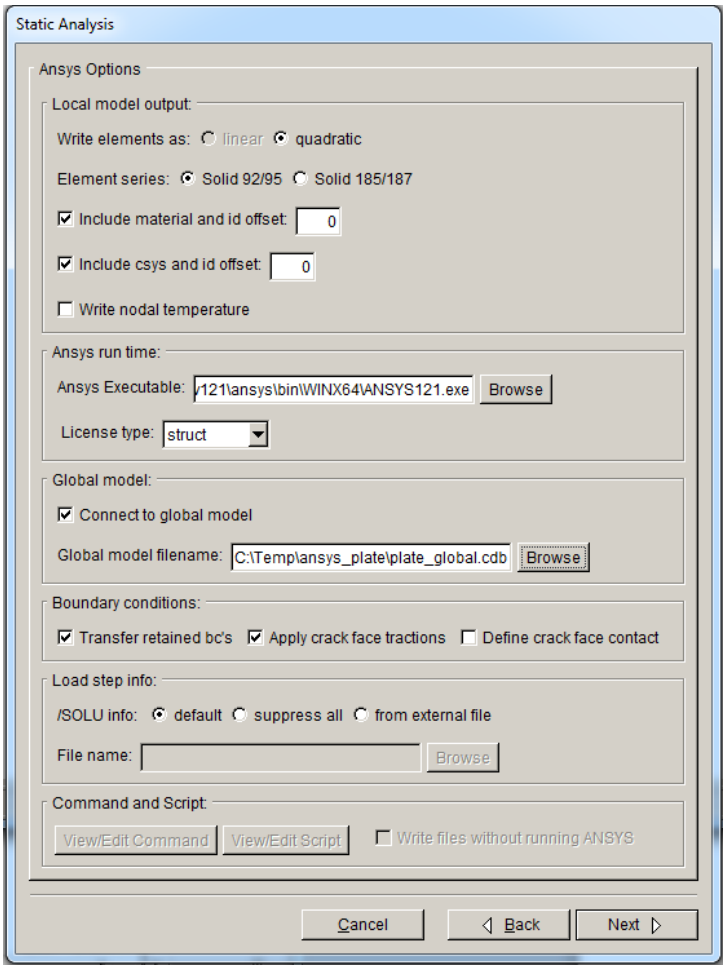

Static Analysis
Ansys Options
Local model output:
Write elements as: linear quadratic
Element series: Solid 92/95 Solid 185/187
Include material and id offset: 0
Include csys and id offset: 0
Write nodal temperature
Ansys run time:
Ansys Executable: v121\ansys\bin\WINX64\ANSYS121.exe Browse
License type: struct
Global model:
Connect to global model
Global model filename: C:\Temp\ansys_plate\plate_global.cdb Browse
Boundary conditions:
Transfer retained bc's Apply crack face tractions Define crack face contact
Load step info:
/SOLU info: default suppress all from external file
File name: Browse
Command and Script:
View/Edit Command View/Edit Script Write files without running ANSYS
Cancel Back Next

Figure 4.13: Mode I SIFs for crack face tractions using M-Integral

Step 6.4: Compute SIFs

Compute the SIFs when ANSYS is done. In the SIF dialog, make sure that the Crack face traction box is checked, Fig 4.14. Note that because we have two load cases, the SIF dialog has extra options for selecting the load case to be plotted. The first load case SIFs should match those computed previously; compare the Mode I SIFs in Fig 4.15 with those in Fig 4.7. You can also plot the SIFs for the second load case. As expected, the values based on crack face tractions are almost identical to those for far-field loading, differing in the 4th decimal place, Fig 4.16.

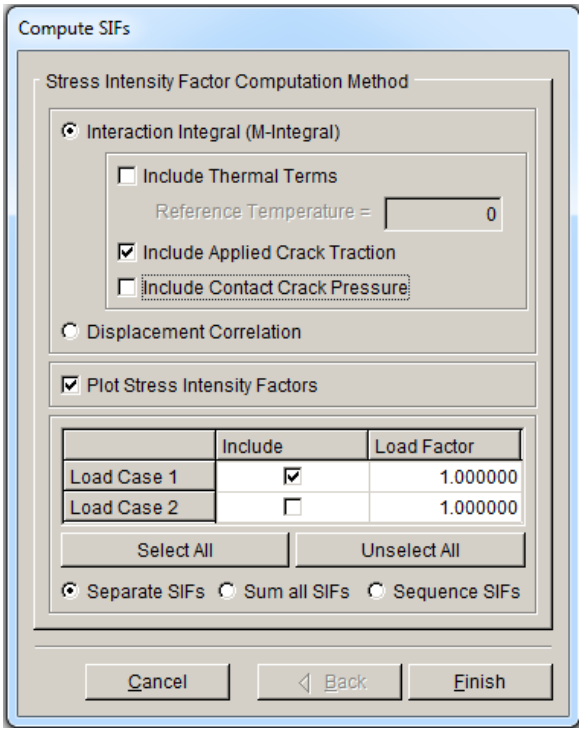


Figure 4.14: Compute SIFs dialog.

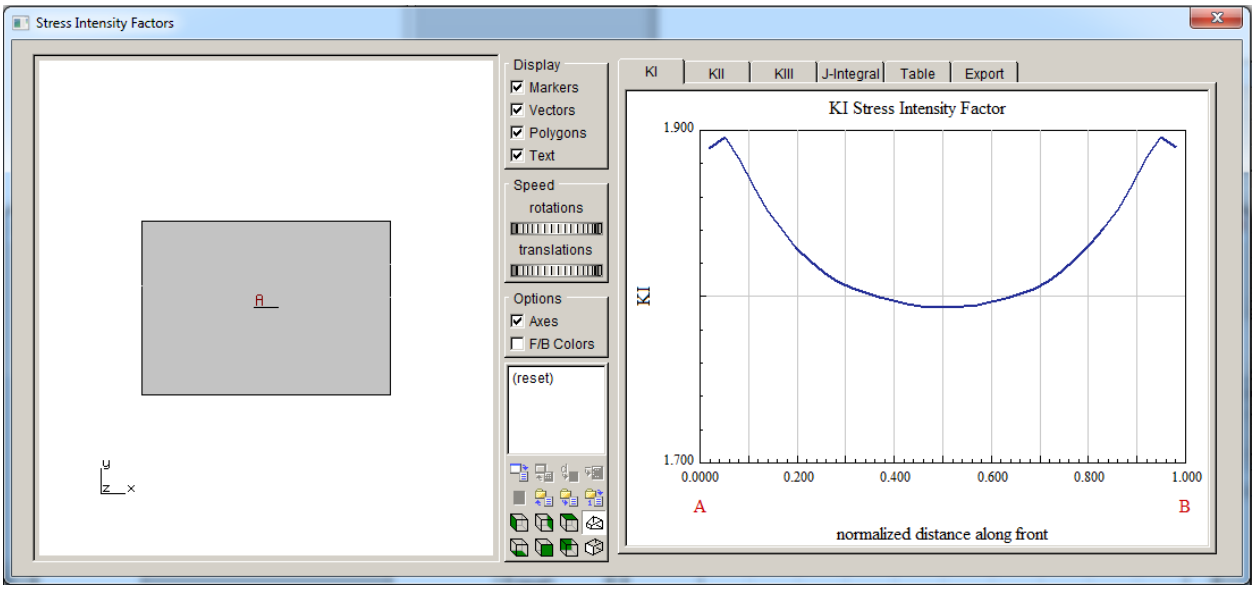


Figure 4.15: Mode I SIFs for crack face tractions using M-Integral

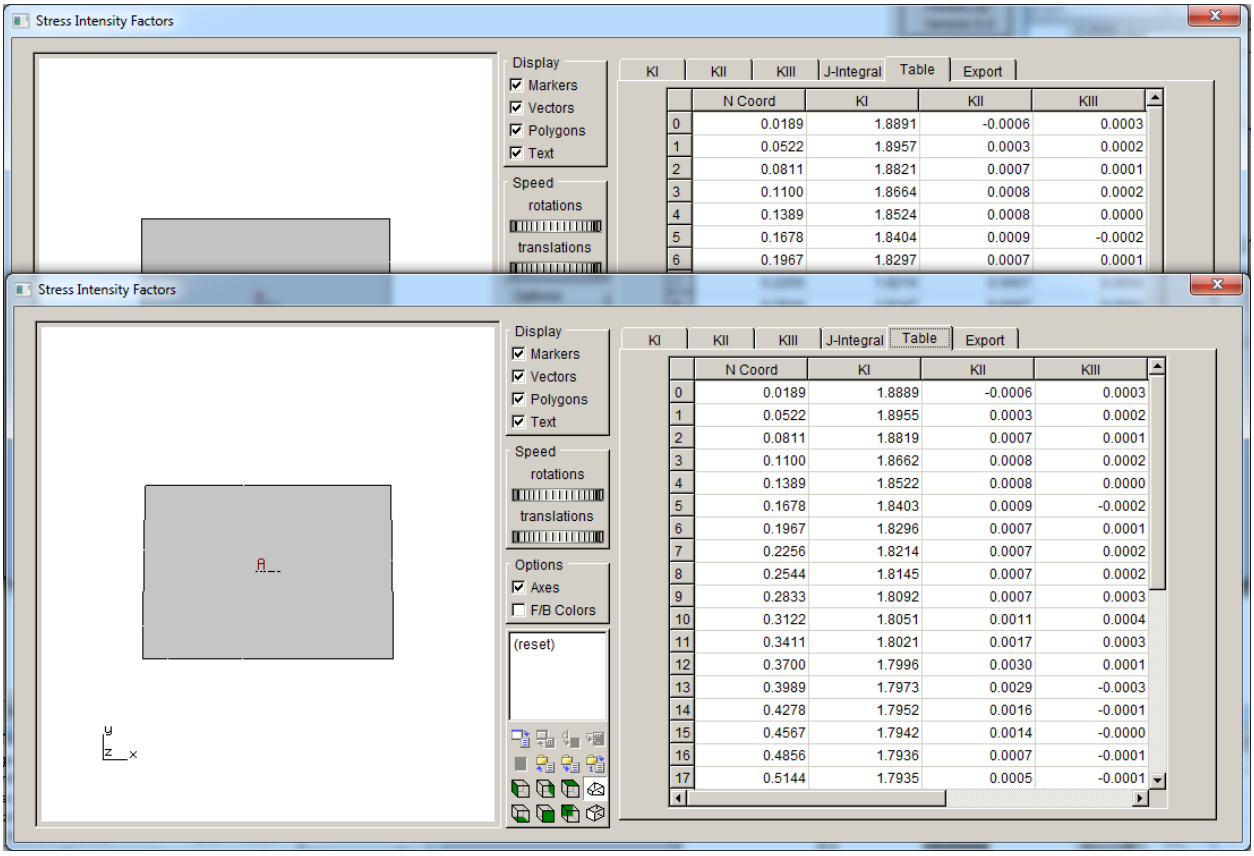


Figure 4.16: SIFs for far-field load (top) compared with those for crack face tractions.

This is the end of the FRANC3D/ANSYS tutorial!